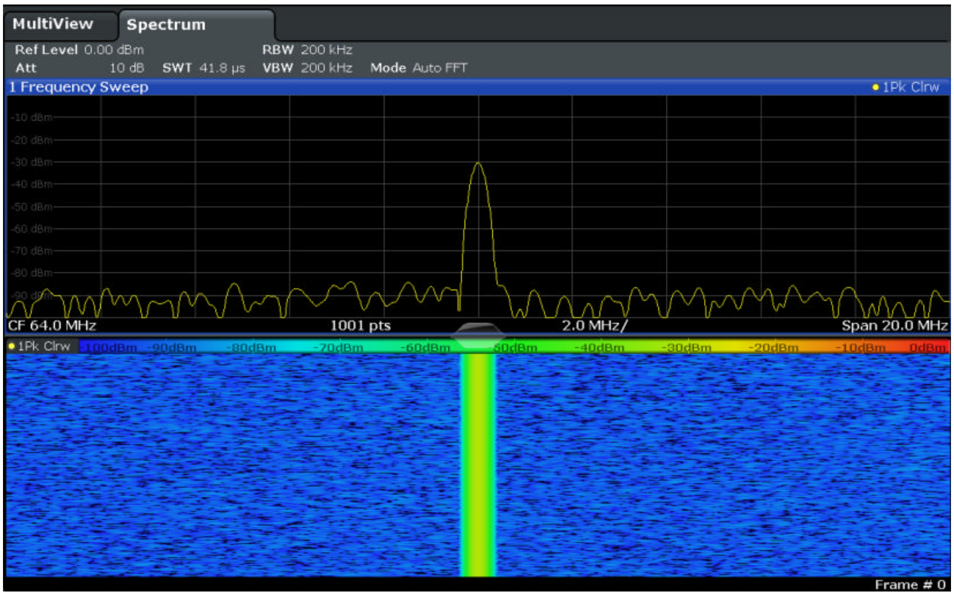
Example: Spectrogram for the calibration signal



In this example you see the spectrogram for the calibration signal of the R&S FSW, compared to the standard spectrum display. Since the signal does not change over time, the color of the frequency levels does not change over time, i.e. vertically. The legend above the spectrogram display describes the power levels the colors represent.

Result display

The spectrogram result can consist of the following elements:

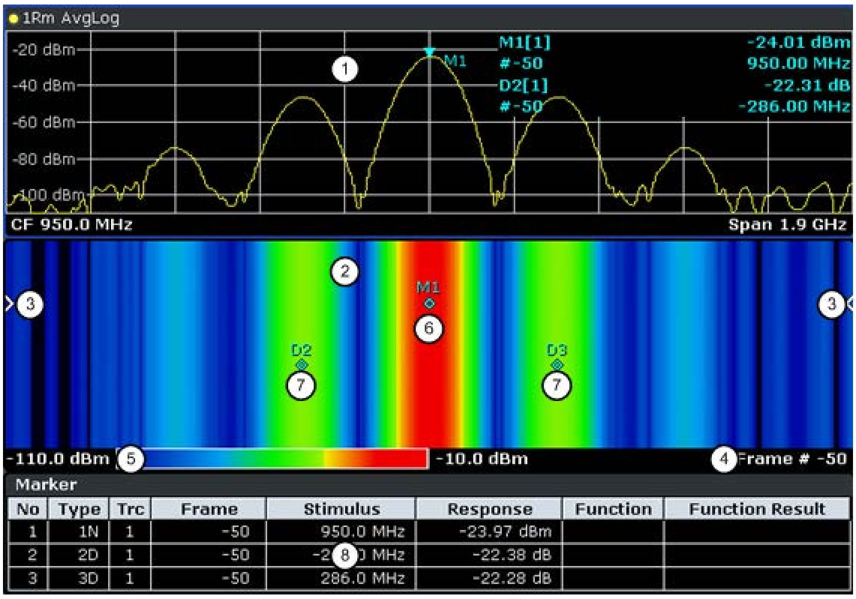


Fig. 6-5: Screen layout of the spectrogram result display

1 = Spectrum result display
2 = Spectrogram result display
3 = Current frame indicator
4 = Time stamp / frame number

5 = Color map
6 = Marker
7 = Delta marker
8 = Marker list

Time Frames

The time information in the spectrogram is displayed vertically, along the y-axis. Each line (or trace) of the y-axis represents one or more captured sweeps and is called a **time frame** or simply "frame". As with standard spectrum traces, several measured values are combined in one sweep point using the selected detector (see chapter 6.3.1.1, "Mapping Samples to Sweep Points with the Trace Detector", on page 237).

Frames are sorted in chronological order, beginning with the most recently recorded frame at the top of the diagram (frame number 0). With the next sweep, the previous frame is moved further down in the diagram, until the maximum number of captured frames is reached. The display is updated continuously during the measurement, and the measured trace data is stored. Spectrogram displays are continued even after single sweep measurements unless they are cleared manually.

The maximum number of frames that you can capture is summarized in table 6-3.

Table 6-3: Correlation between number of sweep points and number of frames stored in the history buffer

Sweep Points	Max. History Depth
≤1250	20000
2001	12488
4001	6247
8.001	3124
16.001	1562
32.001	781

Frame analysis - Frame count vs. sweep count

As described for standard spectrum sweeps, the sweep count defines how many sweeps are analyzed to create a single trace. Thus, for a trace in "Average" mode, for example, a sweep count of 10 means that 10 sweeps are averaged to create a single trace, or frame.

The frame count, on the other hand, determines how many frames are plotted during a single sweep measurement (as opposed to a continuous sweep). For a frame count of 2, for example, 2 frames will be plotted during each single sweep. For continuous sweep mode, the frame count is irrelevant; one frame is plotted per sweep until the measurement is stopped.

If you combine the two settings, 20 sweeps will be performed for each single sweep measurement. The first 10 will be averaged to create the first frame, the next 10 will be averaged to create the second frame.

As you can see, increasing the sweep count increases the accuracy of the individual traces, while increasing the frame count increases the number of traces in the diagram.

Especially for "Average" or "Min hold" and "Max hold" trace modes, the number of sweeps that are analyzed to create a single trace has an effect on the accuracy of the results. Thus, you can also define whether the results from frames in previous traces are considered in the analysis for each new trace ("Continue frame").

Displaying individual frames

The spectrogram diagram includes all stored frames since it was last cleared. Arrows on the left and right border of the spectrogram indicate the currently selected frame. The spectrum diagram always displays the spectrum for the currently selected frame. The current frame number is indicated in the diagram footer, or alternatively a time stamp, if activated. The current frame, displayed at the top of the diagram, is frame number 0. Older frames further down in the diagram are indicated by a negative index, e.g."-10". You can display the spectrum diagram of a previous frame by changing the current frame number.

Color Maps

Spectrograms assign power levels to different colors in order to visualize them. The legend above the spectrogram display describes the power levels the colors represent.

The color display is highly configurable to adapt the spectrograms to your needs. You can define:

- Which colors to use (Color scheme)
- Which value range to apply the color scheme to
- How the colors are distributed within the value range, i.e where the focus of the visualization lies (shape of the color curve

The individual colors are assigned to the power levels automatically by the R&S FSW.

The Color Scheme

You can select which colors are assigned to the measured values. Four different color ranges or "schemes" are available:

- **Hot**

  Uses a color range from blue to red. Blue colors indicate low levels, red colors indicate high ones.
- **Cold**

  Uses a color range from red to blue. Red colors indicate low levels, blue colors indicate high ones.
  The "Cold" color scheme is the inverse "Hot" color scheme.
- **Radar**

  Uses a color range from black over green to light turquoise with shades of green in between. Dark colors indicate low levels, light colors indicate high ones.

- **Grayscale**

  Shows the results in shades of gray. Dark gray indicates low levels, light gray indicates high ones.

The Value Range of the Color Map

If the measured values only cover a small area in the spectrogram, you can optimize the displayed value range so it becomes easier to distinguish between values that are close together, and only parts of interest are displayed at all.

The Shape and Focus of the Color Curve

The color mapping function assigns a specified color to a specified power level in the spectrogram display. By default, colors on the color map are distributed evenly. However, if a certain area of the value range is to be visualized in greater detail than the rest, you can set the focus of the color mapping to that area. Changing the focus is performed by changing the shape of the color curve.

The color curve is a tool to shift the focus of the color distribution on the color map. By default, the color curve is linear. If you shift the curve to the left or right, the distribution becomes non-linear. The slope of the color curve increases or decreases. One end of the color palette then covers a large amount of results, while the other end distributes several colors over a relatively small result range.

You can use this feature to put the focus on a particular region in the diagram and to be able to detect small variations of the signal.

Example:

Fig. 6-6: Linear color curve shape = 0; colors are distributed evenly over the complete result range

In the color map based on the linear color curve, the range from -105.5 dBm to -60 dBm is covered by blue and a few shades of green only. The range from -60 dBm to -20 dBm is covered by red, yellow and a few shades of green.

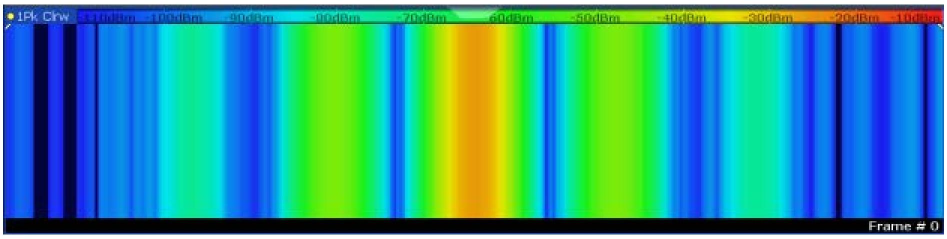


Fig. 6-7: Spectrogram with default color curve

The sample spectrogram is dominated by blue and green colors. After shifting the color curve to the left (negative value), more colors cover the range from -105.5 dBm to -60 dBm (blue, green and yellow), which occurs more often in the example. The range from -60 dBm to -20 dBm, on the other hand, is dominated by various shades of red only.

Fig. 6-8: Non-linear color curve shape = -0.5

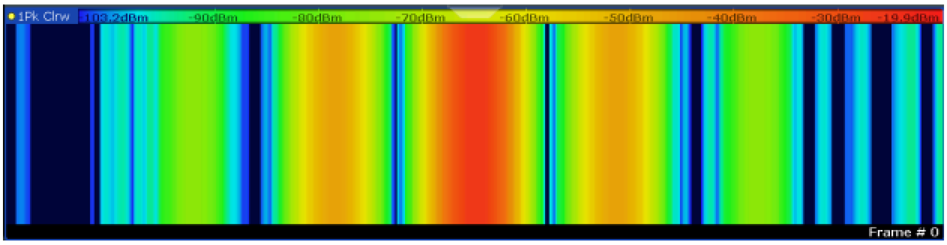


Fig. 6-9: Spectrogram with shifted color curve

Markers in the Spectrogram

Markers and delta markers are shaped like diamonds in the spectrogram. They are only displayed in the spectrogram if the marker position is inside the visible area of the spectrogram. If more than two markers are active, the marker values are displayed in a separate marker table.

In the spectrum result display, the markers and their frequency and level values (1) are displayed as usual. Additionally, the frame number is displayed to indicate the position of the marker in time (2).

In the spectrogram result display, you can activate up to 16 markers or delta markers at the same time. Each marker can be assigned to a different frame. Therefore, in addition to the frequency you also define the frame number when activating a new marker. If no frame number is specified, the marker is positioned on the currently selected frame. All markers are visible that are positioned on a visible frame. Special search functions are provided for spectrogram markers.

In the spectrum result display, only the markers positioned on the currently selected frame are visible. In "Continuous Sweep" mode this means that only markers positioned on frame 0 are visible. To view markers that are positioned on a frame other than frame 0 in the spectrum result display, you must stop the measurement and select the corresponding frame.

6.3.2 Trace Configuration

Trace configuration includes the following settings and functions:

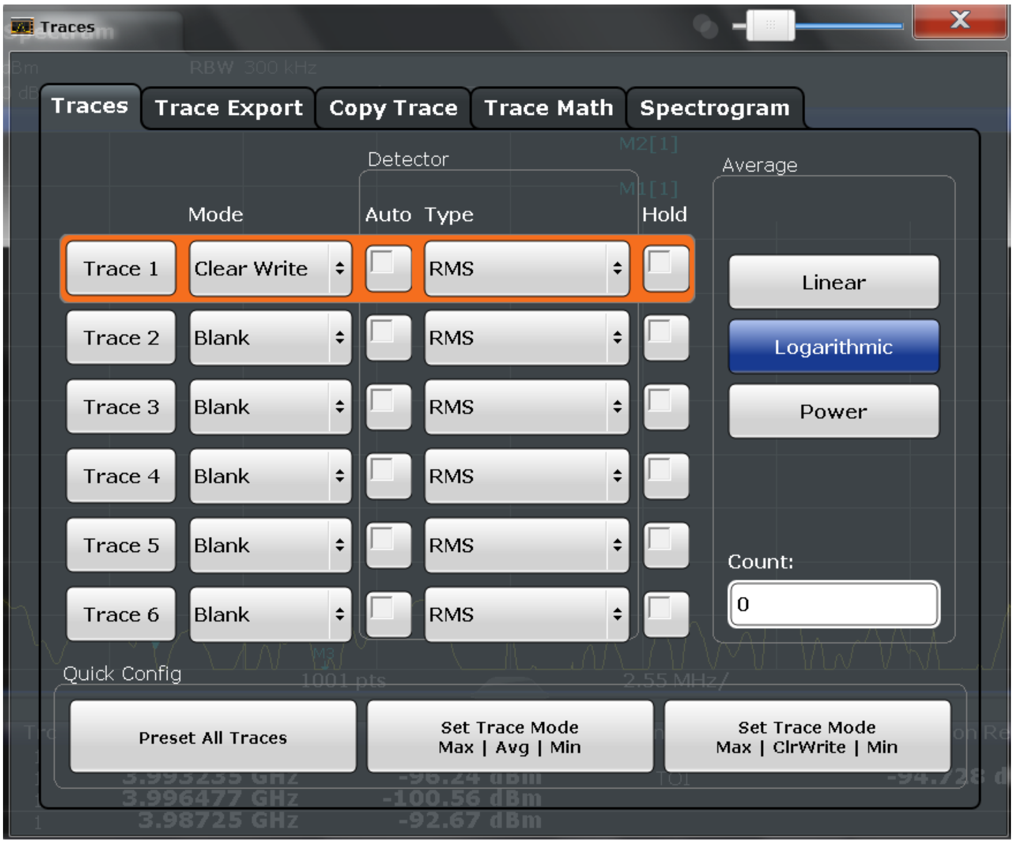
6.3.2.1 Trace Settings

You can configure the settings for up to 6 individual traces.

Trace settings can be configured via the TRACE key, in the "Traces" dialog box, or in the vertical "Traces" tab of the "Analysis" dialog box.

For settings on spectrograms, see chapter 6.3.2.4, "Spectrogram Settings", on page 255.

Trace data can also be exported to an ASCII file for further analysis. For details see chapter 6.3.2.3, "Trace Export Settings", on page 254.



Trace 1/Trace 2/Trace 3/Trace 4/Trace 5/Trace 6

Selects the corresponding trace for configuration. The currently selected trace is highlighted orange.

For details see chapter 6.3.3.1, "How to Configure a Standard Trace", on page 259.

SCPI command:
Selected via numeric suffix of:`TRACe<1...6>` commands

Trace Mode

Defines the update mode for subsequent traces.

For details see chapter 6.3.1.2, "Analyzing Several Traces - Trace Mode", on page 239.

"Clear Write" Overwrite mode: the trace is overwritten by each sweep. This is the default setting.
The "Detector" is automatically set to "Auto Peak".

"Max Hold" The maximum value is determined over several sweeps and displayed. The R&S FSW saves the sweep result in the trace memory only if the new value is greater than the previous one.
The "Detector" is automatically set to "Positive Peak".
This mode is not available for statistics measurements.

"Min Hold" The minimum value is determined from several measurements and displayed. The R&S FSW saves the sweep result in the trace memory only if the new value is lower than the previous one.
The "Detector" is automatically set to "Negative Peak".
This mode is not available for statistics measurements.

"Average" The average is formed over several sweeps.
The Sweep/Average Count determines the number of averaging procedures.
The "Detector" is automatically set to "Sample".
This mode is not available for statistics measurements.

"View" The current contents of the trace memory are frozen and displayed.

"Blank" Removes the selected trace from the display.

SCPI command:
`DISPlay[:WINDow<n>]:TRACe<t>:MODE` on page 613

Detector
Defines the trace detector to be used for trace analysis.

For details see chapter 6.3.1.1, "Mapping Samples to Sweep Points with the Trace Detector", on page 237.

"Auto" Selects the optimum detector for the selected trace and filter mode. This is the default setting.

"Type" Defines the selected detector type.

SCPI command:
`[SENSe:][WINDow:]DETector<trace>[:FUNCtion]` on page 616
`[SENSe:][WINDow:]DETector<trace>[:FUNCtion]:AUTO` on page 616

Hold
If activated, traces in "Min Hold", "Max Hold" and "Average" mode are not reset after specific parameter changes have been made.

Normally, the measurement is started anew after parameter changes, before the measurement results are analyzed (e.g. using a marker). In all cases that require a new measurement after parameter changes, the trace is reset automatically to avoid false results (e.g. with span changes). For applications that require no reset after parameter changes, the automatic reset can be switched off.

The default setting is off.

SCPI command:
`DISPlay[:WINDow<n>]:TRACe<t>:MODE:HCONtinuous` on page 614

Average Mode
Defines the mode with which the trace is averaged over several sweeps. A different averaging mode can be defined for each trace.

This setting is only applicable if trace mode "Average" is selected.

How many sweeps are averaged is defined by the "Sweep/Average Count" on page 203.

For details see chapter 6.3.1.4, "How Trace Data is Averaged - the Averaging Mode", on page 241.

"Linear" The power level values are converted into linear units prior to averaging. After the averaging, the data is converted back into its original unit.

"Logarithmic" For logarithmic scaling, the values are averaged in dBm. For linear scaling, the behavior is the same as with linear averaging.

"Power" Activates linear power averaging.
The power level values are converted into unit Watt prior to averaging. After the averaging, the data is converted back into its original unit.
Use this mode to average power values in Volts or Amperes correctly.

SCPI command:
`[SENSe:]AVERage<n>:TYPE` on page 616

Average Count

Determines the number of averaging or maximum search procedures If the trace modes "Average", "Max Hold" or "Min Hold" are set.

In continuous sweep mode, if sweep count = 0 (default), averaging is performed over 10 sweeps. For sweep count =1, no averaging, maxhold or minhold operations are performed.

This value is identical to the Sweep/Average Count setting in the "Sweep" configuration.

SCPI command:
`[SENSe:]AVERage<n>:COUNt` on page 615

Predefined Trace Settings - Quick Config

Commonly required trace settings have been predefined and can be applied very quickly by selecting the appropriate button.

Function	Trace Settings	
Preset All Traces	Trace 1:	Clear Write Auto Detector (Auto Peak)
	Traces 2-6:	Blank Auto Detector
Set Trace Mode Max | Avg | Min	Trace 1:	Max Hold Auto Detector (Positive Peak)
	Trace 2:	Average Auto Detector (Sample)
	Trace 3:	Min Hold Auto Detector (Negative Peak)
	Traces 4-6:	Blank Auto Detector

Function	Trace Settings	
Set Trace Mode Max | ClrWrite | Min	Trace 1:	Max Hold Auto Detector (Positive Peak)
	Trace 2:	Clear Write Auto Detector (Auto Peak)
	Trace 3:	Min Hold Auto Detector (Negative Peak)
	Traces 4-6:	Blank Auto Detector

Trace 1/Trace 2/Trace 3/Trace 4 (Softkeys)
Displays the "Traces" settings and focuses the "Mode" list for the selected trace.

For details see chapter 6.3.3.1, "How to Configure a Standard Trace", on page 259.

SCPI command:
`DISPlay[:WINDow<n>]:TRACe<t>[:STATe]` on page 615

Copy Trace
The "Copy Trace" softkey opens the "Copy Trace" tab of the "Trace Configuration" dialog box.

The "Copy Trace" tab contains functionality to copy trace data to another trace.

The first group of buttons (labelled "Trace 1" to "Trace 6") select the source trace. The second group of buttons (labelled "Copy to Trace 1" to "Copy to Trace 6") select the destination.

SCPI command:
`TRACe<n>:COPY` on page 617

6.3.2.2 Trace Math

Trace math settings can be configured via the TRACE key, in the "Trace Math" tab of the "Traces" dialog box.

Trace Math Function

Defines which trace is subtracted from trace 1. The result is displayed in trace 1 and refers to the zero point defined with the Trace Math Position setting. The following subtractions can be performed:

"T1-T2 -> T1"	Subtracts trace 2 from trace 1.
"T1-T3 -> T1"	Subtracts trace 3 from trace 1
"T1-T4 -> T1"	Subtracts trace 4 from trace 1
"T1-T5 -> T1"	Subtracts trace 5 from trace 1
"T1-T6 -> T1"	Subtracts trace 6 from trace 1

To switch off the trace math, use the Trace Math Off button.

SCPI command:

`CALCulate<n>:MATH[:EXPression][:DEFine]` on page 622

`CALCulate<n>:MATH:STATe` on page 624

Trace Math Off

Deactivates any previously selected trace math functions.

SCPI command:

`CALC:MATH:STAT OFF`, see `CALCulate<n>:MATH:STATe` on page 624

Trace Math Position

Defines the zero point on the y-axis of the resulting trace in % of the diagram height. The range of values extends from -100 % to +200 %.

SCPI command:
`CALCulate<n>:MATH:POSition` on page 623

Trace Math Mode

Defines the mode for the trace math calculations.

"Lin" Activates linear subtraction, which means that the power level values are converted into linear units prior to subtraction. After the subtraction, the data is converted back into its original unit.
This setting takes effect if the grid is set to a linear scale. In this case, subtraction is done in two ways (depending on the set unit):

- The unit is set to either W or dBm: the data is converted into W prior to subtraction, i.e. averaging is done in W.
- The unit is set to either V, A, dBmV, dBμV, dBμA or dBpW: the data is converted into V prior to subtraction, i.e. subtraction is done in V.

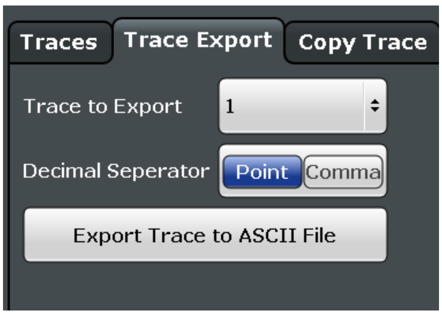
"Log" Activates logarithmic subtraction.
This subtraction method only takes effect if the grid is set to a logarithmic scale, i.e. the unit of the data is dBm. In this case the values are subtracted in dBm. Otherwise (i.e. with linear scaling) the behavior is the same as with linear subtraction.

"Power" Activates linear power subtraction.
The power level values are converted into unit Watt prior to subtraction. After the subtraction, the data is converted back into its original unit. Unlike the linear mode, the subtraction is always done in W.

SCPI command:
`CALCulate<n>:MATH:MODE` on page 623

6.3.2.3 Trace Export Settings

Trace settings can be configured in the "Traces" dialog box or in the vertical "Traces" tab of the "Analysis" dialog box. Switch to the "Trace Export" tab.



Trace to Export
Defines the trace that will be exported to a file.

Decimal Separator
Defines the decimal separator for floating-point numerals for the data export files. Evaluation programs require different separators in different languages.

SCPI command:
`FORMat:DEXPort:DSEParator` on page 678

Export Trace to ASCII File
Opens a file selection dialog box and saves the selected trace in ASCII format (**`.txt`**) to the specified file and directory.

If the spectrogram display is selected when you perform this function, the entire histogram buffer with all frames is exported to a file. The data corresponding to a particular frame begins with information about the frame number and the time that frame was recorded. For large history buffers the export operation may take some time.

For details on the file format see chapter 7.3.4.1, "Reference: ASCII File Export Format", on page 332.

SCPI command:
`MMEMory:STORe<n>:TRACe` on page 697
`MMEMory:STORe:SGRam` on page 697

6.3.2.4 Spectrogram Settings

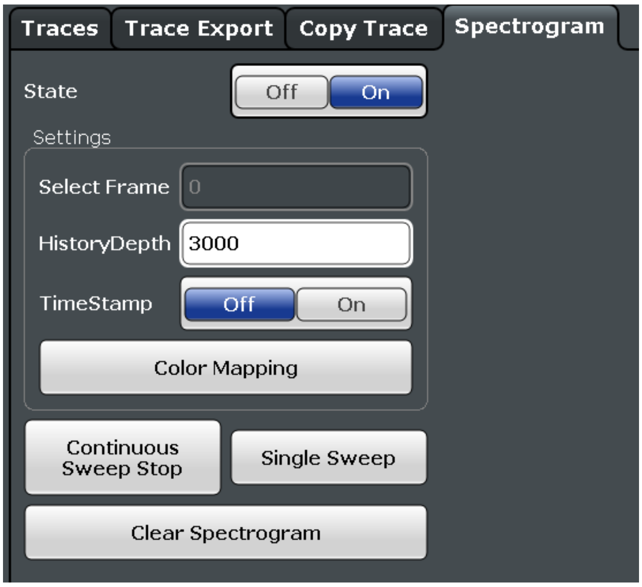
The individual settings available for spectrogram display are described here. For settings on color mapping, see "Color Map Settings" on page 258.

Settings concerning the frames and how they are handled during a sweep are provided as additional sweep settings for spectrogram display, see "Spectrogram Frames" on page 205.

Search functions for spectrogram markers are described in "Marker Search Settings for Spectrograms" on page 283.

General Spectrogram Settings

This section describes general settings for spectrogram display. They are available when you press the TRACE key and then select the "Spectrogram Config" softkey.



State

Activates and deactivates the spectrogram result display

SCPI command:

`CALCulate:SGRam[:STATe]` on page 620
`CALCulate:SGRam:CONT` on page 618

Select frame

Selects a specific frame and loads the corresponding trace from the memory.

Note that activating a marker or changing the position of the active marker automatically selects the frame that belongs to that marker.

This function is available in single sweep mode or if the sweep is stopped.

The most recent frame is number 0, all previous frames have a negative number.

For more information see "Time Frames" on page 244.

SCPI command:

`CALCulate:SGRam:FRAMe:SELect` on page 619

History Depth

Sets the number of frames that the R&S FSW stores in its memory. The maximum number of frames depends on the Sweep Points.

If the memory is full, the R&S FSW deletes the oldest frames stored in the memory and replaces them with the new data.

For an overview of the maximum number of frames depending on the number of sweep points, see table 6-3.

SCPI command:

`CALCulate:SGRam:HDEPth` on page 619

Time Stamp

Activates and deactivates the time stamp. The time stamp shows the system time while the measurement is running. In single sweep mode or if the sweep is stopped, the time stamp shows the time and date of the end of the sweep.

When active, the time stamp replaces the display of the frame number.

SCPI command:

`CALCulate:SGRam:TSTamp[:STATe]` on page 620

`CALCulate:SGRam:TSTamp:DATA?` on page 619

Color Mapping

Opens the "Color Map" dialog.

For details see "Color Maps" on page 245.

Continuous Sweep Stop

Stops a continuous sweep measurement, e.g. in order to display the spectrum display for a previous frame.

Single Sweep/ RUN SINGLE

After triggering, starts the number of sweeps set in "Sweep Count". The measurement stops after the defined number of sweeps has been performed.

While the measurement is running, the "Single Sweep" softkey and the RUN SINGLE key are highlighted. The running measurement can be aborted by selecting the highlighted softkey or key again.

Note: Sequencer. If the Sequencer is active, the "Single Sweep" softkey only controls the sweep mode for the currently selected channel; however, the sweep mode only has an effect the next time the Sequencer activates that channel, and only for a channel-defined sequence. In this case, a channel in single sweep mode is swept only once by the Sequencer.
Furthermore, the RUN SINGLE key on the front panel controls the Sequencer, not individual sweeps. RUN SINGLE starts the Sequencer in single mode.

If the Sequencer is off, only the evaluation for the currently displayed measurement channel is updated.

For details on the Sequencer, see chapter 3.5.1, "The Sequencer Concept", on page 26.

SCPI command:

`INITiate[:IMMediate]` on page 461

Clear Spectrogram

Resets the spectrogram result display and clears the history buffer.

SCPI command:

`CALCulate:SGRam:CLEar[:IMMediate]` on page 618

Color Map Settings

The settings for color mapping are displayed in the "Color Mapping" dialog box that is displayed when you press the "Color Mapping" softkey in the "Spectrogram" menu, or tap the color map in the spectrogram display.

For more information on color maps see "Color Maps" on page 245.

For details on changing color mapping settings see "How to Configure the Color Mapping" on page 262.

In addition to the available color settings, the dialog box displays the current color map and provides a preview of the display with the current settings.

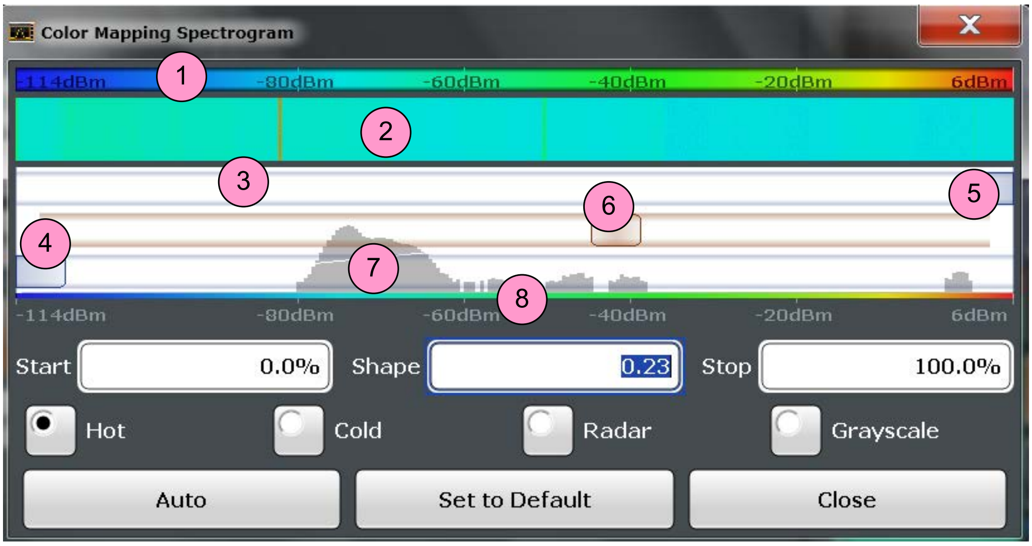


Fig. 6-10: Color Mapping dialog box

1 = Color map: shows the current color distribution
2 = Preview pane: shows a preview of the spectrogram with any changes that you make to the color scheme
3 = Color curve pane: graphical representation of all settings available to customize the color scheme
4/5 = Color range start and stop sliders: define the range of the color map or amplitudes for the spectrogram
6 = Color curve slider: adjusts the focus of the color curve
7 = Histogram: shows the distribution of measured values
8 = Scale of the horizontal axis (value range)

Start / Stop
Defines the lower and upper boundaries of the value range of the spectrogram.

SCPI command:
`DISPlay:WINDow:SGRam:COLor:LOWer` on page 621
`DISPlay:WINDow:SGRam:COLor:UPPer` on page 621

Shape
Defines the shape and focus of the color curve for the spectrogram result display.

"-1 to <0"	More colors are distributed amoung the lower values
"0"	Colors are distributed linearly amoung the values

">0 to 1" More colors are distributed amoung the higher values

SCPI command:
`DISPlay:WINDow:SGRam:COLor:SHAPe` on page 621

Hot/Cold/Radar/Grayscale
Sets the color scheme for the spectrogram.

SCPI command:
`DISPlay:WINDow:SGRam:COLor[:STYLe]` on page 622

Auto
Defines the color range automatically according to the existing measured values for optimized display.

Set to Default
Sets the color mapping to the default settings.

SCPI command:
`DISPlay:WINDow:SGRam:COLor:DEFault` on page 621

6.3.3 How to Configure Traces

The following step-by-step procedures describe the following tasks:

6.3.3.1 How to Configure a Standard Trace

Step-by-step instructions on configuring the trace settings are provided here. For details on individual functions and settings see chapter 6.3.2.1, "Trace Settings", on page 248.

The remote commands required to perform these tasks are described in chapter 10.6.2, "Configuring the Trace Display and Retrieving Trace Data", on page 613.

Trace settings are configured in the "Traces" dialog box.

To display the "Traces" dialog box, do one of the following:

- Press the TRACE key and then select the "Trace Config" softkey.
- Select "Analysis" from the "Overview", then select the "Traces" tab.

1. For each trace, select the "Trace Mode" and "Trace Detector". Traces with the trace mode "Blank" are not displayed.
2. To configure several traces to predefined display modes in one step, press the button for the required function:
   - "Preset All Traces"
   - "Set Trace Mode Avg | Max | Min"
   - "Set Trace Mode Max | ClrWrite | Min"

For details see chapter 6.3.2.1, "Trace Settings", on page 248.

3. For "Average" trace mode, define the number of sweeps to be averaged in the "Sweep/Average Count" field of the "Sweep Config" dialog box.
(Press the SWEEP key and then select the "Sweep Config" softkey to display the "Sweep Config" dialog box.)
4. If linear scaling is used, select the "Average Mode: Linear".
5. To improve the trace stability, increase the number of "Sweep Points" or the "Sweep Time".

   All configured traces (not set to "Blank") are displayed after the next sweep.

How to Copy Traces

1. A trace copy function is provided in a separate tab of the "Traces" dialog box. To display this tab do one of the following:
   - Select the TRACE key and then the "Trace Copy" softkey.
   - Select "Analysis" from the "Overview", then select the "Trace Copy" tab.
2. Select the "Source" trace to be copied.
3. Select the "Copy to trace..." button for the trace to which the settings are to be applied.

   The settings from the source trace are applied to the destination trace. The newly configured trace (if not set to "Blank") is displayed after the next sweep.

6.3.3.2 How to Display and Configure a Spectrogram

Step-by-step instructions on how to display and configure a spectrogram are provided here. For details on individual functions and settings see chapter 6.3.2.4, "Spectrogram Settings", on page 255.

The remote commands required to perform these tasks are described in chapter 10.6.2.2, "Configuring Spectrograms", on page 617.

The following tasks are described here:

- "To display a spectrogram" on page 261
- "To remove the spectrogram display" on page 261
- "To set a marker in the spectrogram" on page 261
- "To configure a spectrogram" on page 261
- "To select a color scheme" on page 262
- "To set the value range graphically using the color range sliders" on page 262
- "To set the value range numerically" on page 263
- "To set the color curve shape graphically using the slider" on page 264
- "To set the color curve shape numerically" on page 264

To display a spectrogram

1. In the "Overview", select "Display", then drag the evaluation type "Spectrogram" to the diagram area.
   Alternatively:
   a) Select the TRACE key and then the "Spectrogram Config" softkey.
   b) Toggle "Spectrogram" to "ON".
2. To clear an existing spectrogram display, select "Clear Spectrogram".
3. Start a new measurement using RUN SINGLE or RUN CONT.
   The spectrogram is updated continuously with each new sweep.
4. To display the spectrum diagram for a specific time frame:
   a) Stop the continuous measurement or wait until the single sweep is completed.
   b) Select the frame number in the diagram footer.
   c) Enter the required frame number in the edit dialog box.
   Note that the most recent sweep is frame number 0, all previous frames have negative numbers.

To remove the spectrogram display

1. Select the TRACE key and then the "Spectrogram Config" softkey.
2. Toggle "Spectrogram" to "OFF".
   The standard spectrum display is restored.

To set a marker in the spectrogram

1. While a spectrogram is displayed, select the MARKER key.
2. Select a "Marker" softkey.
3. Enter the frequency or time (x-value) of the marker or delta marker.
4. Enter the frame number for which the marker is to be set, for example *0* for the current frame, or *-2* for the second to last frame. Note that the frame number is always 0 or a negative value!
   The marker is only visible in the spectrum diagram if it is defined for the currently selected frame. In the spectrogram result display all markers are visible that are positioned on a visible frame.

To configure a spectrogram

1. Configure the spectrogram frames:
   a) Select the SWEEP key.
   b) Select the "Sweep Config" softkey.
   c) In the "Sweep/Average Count" field, define how many sweeps are to be analyzed to create a single frame.
   d) In the "Frame Count" field, define how many frames are to be plotted during a single sweep measurement.

e) To include frames from previous sweeps in the analysis of the new frame (for "Max Hold", "Min Hold" and "Average" trace modes only), select "Continue Frame" = "ON".

2. Define how many frames are to be stored in total:
   a) Select the TRACE key and then the "Spectrogram Config" softkey.
   b) Select the "History Depth" softkey.
   c) Enter the maximum number of frames to store.
3. Optionally, replace the frame number by a time stamp by toggling the "Timestamp" softkey to "On".
4. If necessary, adapt the color mapping for the spectrogram to a different value range or color scheme as described in "How to Configure the Color Mapping" on page 262.

How to Configure the Color Mapping

The color display is highly configurable to adapt the spectrograms to your needs.

The settings for color mapping are defined in the "Color Mapping" dialog box. To display this dialog box, do one of the following:

- Tap the color map in the spectrogram display.
- Press the "Color Mapping" softkey in the "Spectrogram" menu.

To select a color scheme

You can select which colors are assigned to the measured values.

► In the "Color Mapping" dialog box, select the option for the color scheme to be used.

Editing the value range of the color map

The distribution of the measured values is displayed as a histogram in the "Color Mapping" dialog box (see "Color Map Settings" on page 258). To cover the entire measurement value range, make sure the first and last bar of the histogram are included. To remove noise from the display, exclude the bottom 10 or 20 dB of the histogram.

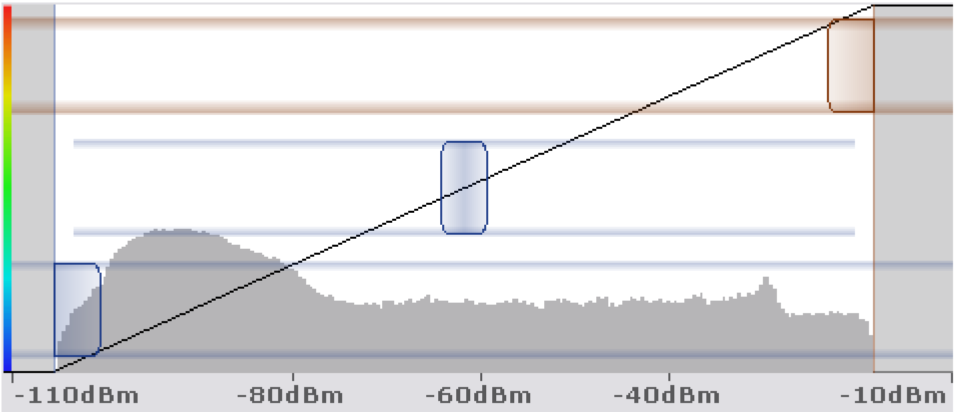
The value range of the color map must cover at least 10% of the value range on the horizontal axis of the diagram.

The value range can be set numerically or graphically.

To set the value range graphically using the color range sliders

1. Select and drag the bottom color curve slider (indicated by a gray box at the left of the color curve pane) to the lowest value you want to include in the color mapping.

2. Select and drag the top color curve slider (indicated by a gray box at the right of the color curve pane) to the highest value you want to include in the color mapping.



To set the value range numerically

1. In the "Start" field, enter the percentage from the left border of the histogram that marks the beginning of the value range.
2. In the "Stop" field, enter the percentage from the right border of the histogram that marks the end of the value range.

Example:

The color map starts at -100 dBm and ends at 0 dBm (i.e. a range of 100 dB). In order to suppress the noise, you only want the color map to start at -90 dBm. Thus, you enter *10%* in the "Start" field. The R&S FSW shifts the start point 10% to the right, to -90 dBm.

Adjusting the reference level and level range

Note that changing the reference level and level range of the measurement also affects the color mapping in the spectrogram.

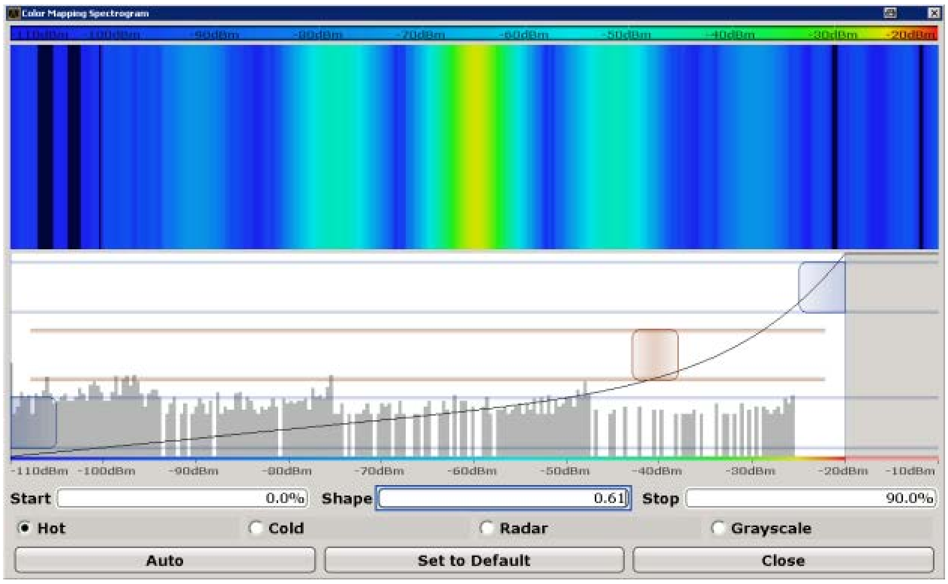
Editing the shape of the color curve

The color curve is a tool to shift the focus of the color distribution on the color map. By default, the color curve is linear, i.e. the colors on the color map are distributed evenly. If you shift the curve to the left or right, the distribution becomes non-linear. The slope of the color curve increases or decreases. One end of the color palette then covers a large amount of results, while the other end distributes several colors over a relatively small result range.

The color curve shape can be set numerically or graphically.

To set the color curve shape graphically using the slider

► Select and drag the color curve shape slider (indicated by a gray box in the middle of the color curve) to the left or right. The area beneath the slider is focussed, i.e. more colors are distributed there.



To set the color curve shape numerically

► In the "Shape" field, enter a value to change the shape of the curve:

- A negative value (-1 to <0) focusses the lower values
- 0 defines a linear distribution
- A positive value (>0 to 1) focusses the higher values

6.3.3.3 How to Copy Traces

You can copy the trace settings from one trace to another in the "Copy Trace" tab of the "Traces" dialog box.

► Select the "Source" trace and then the button for the "Copy to" trace.

SCPI command:

`TRACe<n>:COPY` on page 617

6.4 Marker Usage

Markers help you analyze your measurement results by determining particular values in the diagram. Thus you can extract numeric values from a graphical display both in the time and frequency domain. In addition to basic markers, sophisticated marker functions are provided for special results such as noise or demodulation.

Markers in Spectrogram Displays

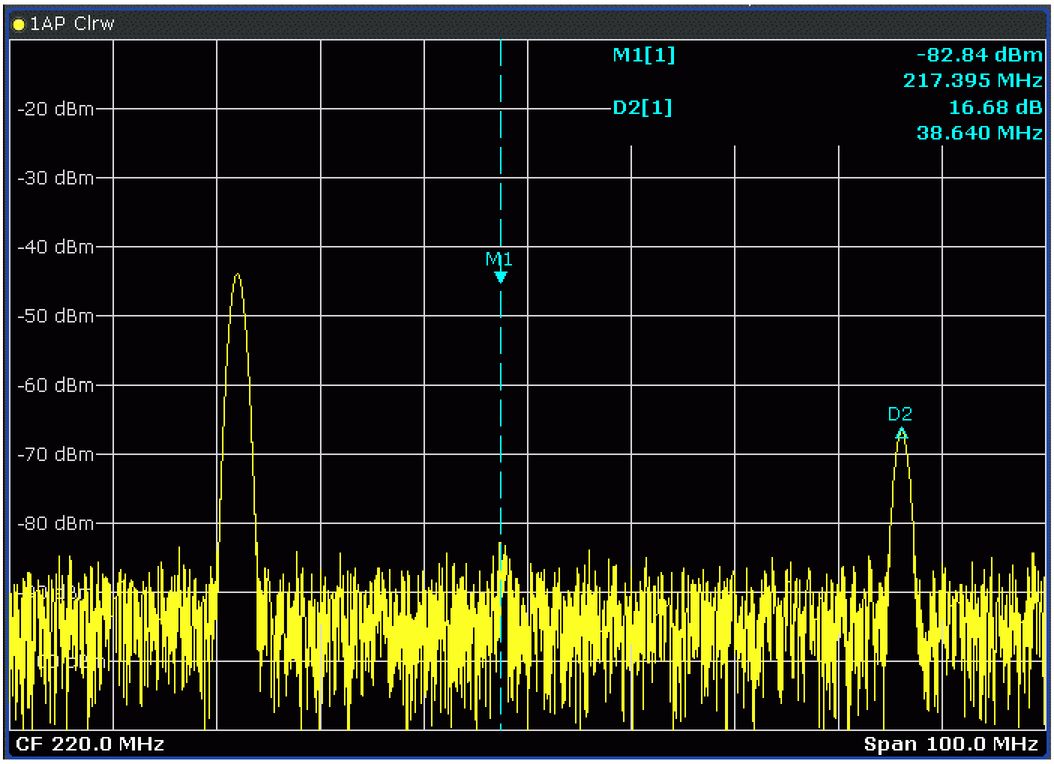
In the spectrogram result display, you can activate up to 16 markers or delta markers at the same time. Each marker can be assigned to a different frame. Therefore, in addition to the frequency you also define the frame number when activating a new marker. If no frame number is specified, the marker is positioned on the currently selected frame. All markers are visible that are positioned on a visible frame.

6.4.1 Basics on Markers and Marker Functions

Some background knowledge on marker settings and functions is provided here for a better understanding of the required configuration settings.

Markers are used to mark points on traces, to read out measurement results and to select a display section quickly. R&S FSW provides 16 markers per display window. In the Spectrum application, the same markers are displayed in all windows.

- The easiest way to work with markers is using the touch screen. Simply drag the marker and drop it at the required position. When a marker label is selected, a vertical line is displayed which indicates the marker's current x-value.



- Alternatively, change the position of the selected marker using the rotary knob. By default, the marker is moved from one pixel to the next. If you need to position the marker more precisely, change the step size to move from one sweep point to the next (General Marker Setting).
- You can also set an active marker to a new position by defining its x-position numerically. When you select the softkey for a marker, an edit dialog box is displayed.
- The most commonly required marker settings and functions are also available as softkeys or via the context menu. Tap the marker on the touch screen and hold your finger for about 2 seconds until the context menu is opened, then select the required entry.
- Softkeys for active markers (displayed on the screen) are highlighted blue. The softkey for the currently selected marker (for which functions are performed) is highlighted orange.
- To set individual markers very quickly, use the softkeys in the "Marker" menu.
- To set up several markers at once, use the "Marker" dialog box.
- To position the selected marker to a special value, use the softkeys in the "Marker To" menu.
- To determine more sophisticated marker results, use the special functions in the "Marker Function" dialog box.

6.4.1.1 Marker Types

All markers can be used either as normal markers or delta markers. A normal marker indicates the absolute signal value at the defined position in the diagram. A delta marker indicates the value of the marker relative to the specified reference marker (by default marker 1).

In addition, special functions can be assigned to the individual markers. The availability of special marker functions depends on whether the measurement is performed in the frequency or time domain.

Temporary markers are used in addition to the markers and delta markers to analyze the measurement results for special marker functions. They disappear when the associated function is deactivated.

6.4.1.2 Activating Markers

Only active markers are displayed in the diagram and in the marker table. Active markers are indicated by a highlighted softkey.

By default, marker 1 is active and positioned on the maximum value (peak) of trace 1 as a normal marker. If several traces are displayed, the marker is set to the maximum value of the trace which has the lowest number and is not frozen (View mode). The next marker to be activated is set to the frequency of the next lower level (next peak) as a delta marker; its value is indicated as an offset to marker 1.

A marker can only be activated when at least one trace in the corresponding window is visible. If a trace is switched off, the corresponding markers and marker functions are also deactivated. If the trace is switched on again, the markers along with coupled functions are restored to their original positions, provided the markers have not been used on another trace.

6.4.1.3 Marker Results

Normal markers point to a sweep point on the time or frequency axis and display the associated numeric value for that sweep point. delta markers indicate an offset between the level at the delta marker position and the level at the position of the assigned reference marker, in dB. Signal count markers determine the frequency of a signal at the marker position very accurately.

The results can be displayed directly within the diagram area or in a separate table. By default, the first two active markers are displayed in the diagram area. If more markers are activated, the results are displayed in a marker table.

Marker information in diagram area

By default, the results of the last two markers or delta markers that were activated are displayed in the diagram area.

The following information is displayed there:

- The marker type (M for normal, D for delta, or special function name)
- The marker number (1 to 16)
- The assigned trace number in square brackets [ ]
- The marker value (response) on the y-axis, or the result of the marker function
- The marker position (stimulus) on the x-axis

For n dB down markers, additional information is displayed, see "Measuring Characteristic Bandwidths (n dB Down Marker)" on page 273.

Marker information in marker table

In addition to the marker information displayed within the diagram area, a separate marker table may be displayed beneath the diagram. This table provides the following information for all active markers:

Type	Marker type: N (normal), D (delta), T (temporary, internal) and number
Dgr	Diagram number
Ref	Reference marker for delta markers
Trc	Trace to which the marker is assigned
Stimulus	x-value of the marker
Response	y-value of the marker
Function	Activated marker or measurement function
Function Result	Result of the active marker or measurement function

6.4.1.4 Searching for Signal Peaks

A common task in spectrum analysis is determining peak values, i.e. maximum or minimum signal levels. The R&S FSW provides various peak search functions and applications:

- Setting a marker to a peak value once (Peak Search)
- Searching for a peak value within a restricted search area (Search Limits)
- Creating a marker table with all or a defined number of peak values for one sweep (Marker Peak List)
- Updating the marker position to the current peak value automatically after each sweep (Auto Peak Search)
- Creating a fixed reference marker at the current peak value of a trace (Peak Search)

Peak search limits

The peak search can be restricted to a search area. The search area is defined by limit lines which are also indicated in the diagram. In addition, a minimum value (threshold) can be defined as a further search condition.

When is a peak a peak? - Peak excursion

During a peak search, for example when a marker peak table is displayed, noise values may be detected as a peak if the signal is very flat or does not contain many peaks. Therefore, you can define a relative threshold ("Peak excursion"). The signal level must increase by the threshold value before falling again before a peak is detected. To avoid identifying noise peaks as maxima or minima, enter a peak excursion value that is higher than the difference between the highest and the lowest value measured for the displayed inherent noise.

Effect of peak excursion settings (example)

The following figure shows a trace to be analyzed.

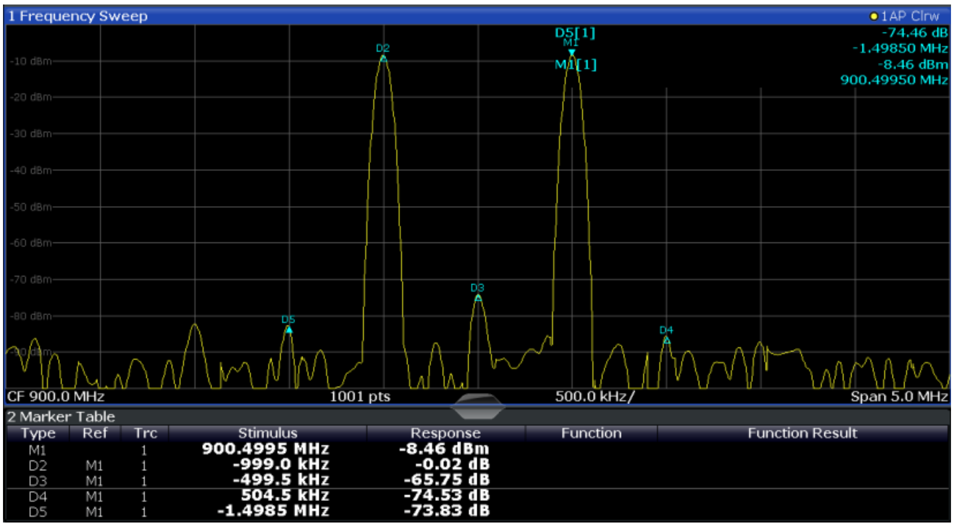


Fig. 6-11: Trace example

The following table lists the peaks as indicated by the marker numbers in the diagram above, as well as the minimum decrease in amplitude to either side of the peak:

Marker #	Min. amplitude decrease to either side of the signal
1	30 dB
2	29.85 dB
3	20 dB
4	10 dB
5	18 dB

In order to eliminate the smaller peaks M3,M4 and M5 in the example above, a peak excursion of at least 20 dB is required. In this case, the amplitude must rise at least 20 dB before falling again before a peak is detected.

Marker peak list

The marker peak list determines the frequencies and levels of peaks in the spectrum. It is updated automatically after each sweep. How many peaks are displayed can be defined, as well as the sort order. In addition, the detected peaks can be indicated in the diagram. The peak list can also be exported to a file for analysis in an external application.

Automatic peak search

A peak search can be repeated automatically after each sweep in order to keep the maximum value as the reference point for a phase noise measurement. This is useful to track a drifting source. The delta marker 2, which shows the phase noise measurement result, keeps the delta frequency value. Therefore the phase noise measurement leads to reliable results in a certain offset although the source is drifting.

Using a peak as a fixed reference marker

Some results are analyzed in relation to a peak value, for example a carrier frequency level. In this case, the maximum level can be determined by an initial peak search and then be used as a reference point for further measurement results.

6.4.1.5 Special Marker Functions

In addition to basic markers, sophisticated marker functions are provided for special results such as noise or demodulation.

Performing a Highly Accurate Frequency Measurement (Signal Count)

A normal marker determines the position of the point on the trace and indicates the signal frequency at this position. The trace, however, contains only a limited number of points. Depending on the selected span, each trace point may contain many measurement values. Thus, the frequency resolution of each trace point is limited (see also chapter 5.5.1.8, "How Much Data is Measured: Sweep Points and Sweep Count", on page 198). Frequency resolution is further restricted by the RBW and sweep time settings.

In order to determine the frequency of a signal point accurately without changing the sweep settings, the R&S FSW is equipped with a signal counter. The signal counter sets the RF to the current marker position, then counts the zero crossings of the IF (thus the term signal *counter*) and derives the precise frequency value.

Signal counting can be performed explicitely at the current marker position ("Signal Count" marker function), or implicitely by the R&S FSW for certain functions.

Signal counting is only possible while the instrument is not sweeping. Thus, to perform a signal count for a marker, the sweep is stopped at the marker position. The frequency is determined with the desired resolution and then the sweep is allowed to continue.

Measuring Noise Density

Using the noise measurement marker function, the noise power density is measured at the position of the marker. In the time domain mode, all points of the trace are used to determine the noise power density. When measurements are performed in the frequency domain, two points to the right and left of the marker are used for the measurement to obtain a stable result.

Noise density is the noise referred to a bandwidth of 1 Hz. With logarithmic amplitude units (dBm, dBmV, dBmμV, dBμA), the noise power density is output in dBm/Hz, i.e. as level in 1 Hz bandwidth with reference to 1 mW. With linear amplitude units (V, A, W), the noise voltage density is analyzed in μV/Hz, the noise current density in μA/Hz or the noise power density in μW/Hz. The result is indicated as the noise marker value.

Prerequisite settings

The following settings have to be made to obtain correct values:

- Detector: Sample or RMS
- Video bandwidth:
  ≤ 0.1 resolution bandwidth with sample detector
  ≥ 3 x resolution bandwidth with RMS detector
- Trace averaging:
  In the default setting, the R&S FSW uses the sample detector for the noise function. With the sample detector, the trace can additionally be set to "Average" mode to stabilize the measured values. When the RMS detector is used, trace averaging should not be used since in this case it produces too low noise levels which cannot be corrected. Instead, the sweep time can be increased to obtain stable measurement results.

Correction factors

The R&S FSW uses the following correction factors to analyze the noise density from the marker level:

- Since the noise power is indicated with reference to 1 Hz bandwidth, the bandwidth correction value is deducted from the marker level. It is 10 x lg (1 Hz/BWNoise), where BWNoise is the noise or power bandwidth of the set resolution filter (RBW).
- RMS detector: With the exception of bandwidth correction, no further corrections are required since this detector already indicates the power for each point of the trace.
- Sample detector: As a result of video filter averaging and trace averaging, 1.05 dB is added to the marker level. This is the difference between the average value and the RMS value of white noise. With a logarithmic level axis, 1.45 dB is added additionally. Logarithmic averaging is thus fully taken into account which yields a value that is 1.45 dB lower than that of linear averaging.

- To allow a more stable noise display the adjacent (symmetric to the measurement frequency) points of the trace are averaged.
- For span > 0, the measured values are averaged versus time (after a sweep).

The R&S FSW noise figure can be calculated from the measured power density level. It is calculated by deducting the set RF attenuation (RF Att) from the displayed noise level and adding 174 to the result.

Measuring Phase Noise

Phase noise is unintentional modulation of a carrier; it creates frequencies next to the carrier frequency. A phase noise measurement consists of noise density measurements at defined offsets from the carrier; the results are given in relation to the carrier level (dBc). The phase noise marker function measures the noise power at the delta markers referred to 1 Hz bandwidth. Marker 1 is used as the reference for the phase noise measurement. By default, the current frequency and level of marker 1 are used as the fixed reference marker. However, a peak search can be started to use the current signal peak as the reference point, or a reference point can be defined manually.

Since the reference point is fixed, the reference level or the center frequency can be set so that the carrier is outside the displayed frequency range after phase noise measurement is started. Or a notch filter can be switched on to suppress the carrier.

Alternatively, the reference point can be determined automatically by a peak search after each sweep. This function can be used to track a drifting source during a phase noise measurement. The delta marker 2, which shows the phase noise measurement result, keeps the delta frequency value. Therefore the phase noise measurement leads to reliable results in a certain offset although the source is drifting. Only if the marker 2 reaches the border of the span, the delta marker value is adjusted to be within the span. In these cases, select a larger span.

The result of the phase noise measurement is the difference in level between the reference point and the noise power density. It is indicated as the function result of the phase noise marker.

The sample detector is automatically used and the video bandwidth set to 0.1 times the resolution bandwidth (RBW). The two settings are taken into account in the correction values used for the noise power measurement. To obtain stable results, two pixels on the right and the left of the delta marker position are taken for the measurement. The procedure for determining the noise power is identical to the method used for the noise power measurement (see "Measuring Noise Density" on page 271).

Using logarithmic scaling for the frequency axis allows for a large frequency range with fine resolution close to the carrier.

Defining a Fixed Reference Marker

Instead of using a reference marker that may vary its position depending on the measurement results, a fixed reference marker can be defined for trace analysis. Once positioned, the reference marker does not move during subsequent sweeps unless you explicitely move it manually.

Measuring Characteristic Bandwidths (n dB Down Marker)

When characterizing the shape of a signal, the bandwidth at a specified offset from its peak level is often of interest. The offset is specified as a relative decrease in amplitude of n dB. In order to measure this bandwidth, you could use several markers and delta markers and determine the bandwidth manually. However, using the n dB down marker function makes the task very simple and quick.

The n dB down marker function uses the current value of marker 1 as the reference point. It activates two temporary markers T1 and T2 located on the signal, whose level is n dB below the level of the reference point. Marker T1 is placed to the left and marker T2 to the right of the reference marker. The default setting for n is 3 dB, but it can be changed.

If a positive offset is entered, the markers T1 and T2 are placed below the active reference point. If a negative value is entered (for example for notch filter measurements), the markers T1 and T2 are placed above the active reference point.

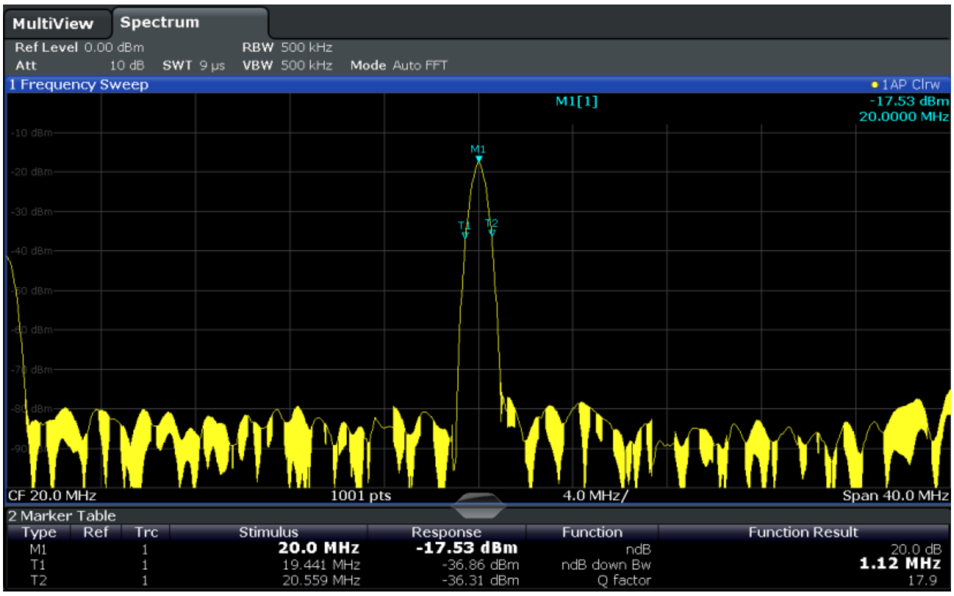


Fig. 6-12: n dB down marker function

The following marker function results are displayed:

Table 6-4: n dB down marker function results

Label	Description
M1	Current position and level of marker 1
ndB	Offset value (n dB down)
ndB down Bw / PWid	Determined bandwidth or pulse width (zero span) at the offset
Q-factor	Quality factor of the determined bandwidth (characteristic of damping or resonance)
T1, T2	Current position and level of the temporary markers

If the required position for the temporary markers cannot be determined uniquely, for example due to noise, dashes are displayed as a result.

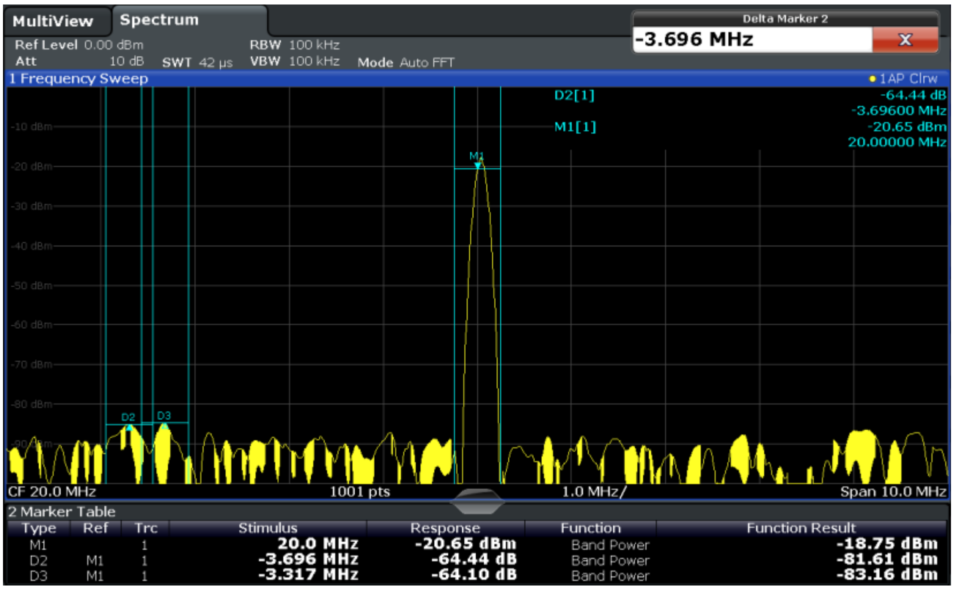
Measuring the Power in a Channel (Band)

When you want to determine the noise power in a transmission channel, you could use a noise marker and multiply the result with the channel bandwidth. However, the results would only be accurate for flat noise.

Band power markers allow you to measure the integrated power for a defined span (band) around a marker (similar to ACP measurements). By default, 5 % of the current span is used. The span is indicated by limit lines in the diagram. The results can be displayed either as a power (dBm) or density (dBm/Hz) value and are indicated in the marker table for each band power marker.

Band power markers are only available for standard frequency measurements (not zero span) in the Spectrum application.

The entire band must lie within the display. If it is moved out of the display, the result cannot be calculated (indicated by "- - -" as the "Function Result"). However, the width of the band is maintained so that the band power can be calculated again when it returns to the display.



All markers can be defined as band power markers, each with a different span. When a band power marker is activated, if no marker is active yet, marker 1 is activated. Otherwise, the currently active marker is used as a band power marker (all other marker functions for this marker are deactivated).

If the detector mode for the marker trace is set to "Auto", the RMS detector is used.

Demodulating Marker Values and Providing Audio Output

The R&S FSW provides demodulators for AM, FM and PM signals. The demodulation marker function sends the demodulated data at the current marker frequency to the audio output. Thus, a displayed signal can be identified acoustically through the use of the internal loudspeaker or with headphones.

This function is not available for Spectrum Emission Mask measurements.

The sweep stops at the frequency determined by marker 1 for the selected time and the RF signal is demodulated in a bandwidth that corresponds to the RBW. Alternatively, demodulation can be activated continuously, i.e. audio output occurs regardless of the marker position and the marker stop time. For measurements in the time domain (zero span), demodulation is always continuous.

Optionally, a mimumum level ("Squelch level") can be defined so that the signal is only demodulated when it exceeds the set level. This is useful during continuous demodulation to avoid listening to noise.

The squelch function activates the video trigger function (see "Video" on page 217) and deactivates any other trigger or gating settings. The squelch level and trigger level are set to the same value. The trigger source in the channel bar is indicated as "SQL" for squelch. The squelch level is indicated by a red line in the diagram.

6.4.2 Marker Configuration

When working with markers, the following configuration settings and functions are available:

6.4.2.1 Marker Settings

Marker settings can be configured via the MARKER key or in the "Marker" dialog box. To display the "Marker" dialog box, do one of the following:

● Press the MKR key, then select the "Marker Config" softkey.
● In the "Overview", select "Analysis", and switch to the vertical "Marker" tab.

The remote commands required to define these settings are described in chapter 10.6.3.1, "Setting Up Individual Markers", on page 628.

Individual Marker Setup

Up to 17 markers or delta markers can be activated for each window simultaneously. Initial marker setup is performed using the "Marker" dialog box.

The markers are distributed among 3 tabs for a better overview. By default, the first marker is defined as a normal marker, whereas all others are defined as delta markers with reference to the first marker. All markers are assigned to trace 1, but only the first marker is active.

Selected Marker

Marker name. The marker which is currently selected for editing is highlighted orange.

SCPI command:

Marker selected via suffix <m> in remote commands.

Marker State

Activates or deactivates the marker in the diagram.

SCPI command:

`CALCulate<n>:MARKer<m>[:STATe]` on page 631

`CALCulate<n>:DELTamarker<m>[:STATe]` on page 630

Marker Position (Stimulus)

Defines the position (x-value) of the marker in the diagram.

SCPI command:

`CALCulate<n>:MARKer<m>:X` on page 632

`CALCulate<n>:DELTamarker<m>:X` on page 630

Frame (Spectrogram only)

Spectrogram frame the marker is assigned to.

SCPI command:

`CALCulate:MARKer<m>:SGRam:FRAMe` on page 644

Marker Type

Toggles the marker type.

The type for marker 1 is always "Normal", the type for delta marker 1 is always "Delta". These types cannot be changed.

Note: If normal marker 1 is the active marker, switching the "Mkr Type" activates an additional delta marker 1. For any other marker, switching the marker type does not activate an additional marker, it only switches the type of the selected marker.

"Normal"	A normal marker indicates the absolute value at the defined position in the diagram.
"Delta"	A delta marker defines the value of the marker relative to the specified reference marker (marker 1 by default).

SCPI command:

`CALCulate<n>:MARKer<m>[:STATe]` on page 631

`CALCulate<n>:DELTamarker<m>[:STATe]` on page 630

Reference Marker

Defines a marker as the reference marker which is used to determine relative analysis results (delta marker values).

If a fixed reference point is configured (see "Defining a Fixed Reference" on page 279), the reference point ("FXD") can also be selected instead of another marker.

SCPI command:

`CALCulate<n>:DELTamarker<m>:MREF` on page 629

Linking to Another Marker

Links the current marker to the marker selected from the list of active markers. If the x-axis value of the inital marker is changed, the linked marker follows on the same x-position. Linking is off by default.

Using this function you can set two markers on different traces to measure the difference (e.g. between a max hold trace and a min hold trace or between a measurement and a reference trace).

SCPI command:

`CALCulate<n>:MARKer<m1>:LINK:TO:MARKer<m2>` on page 631

`CALCulate<n>:DELTamarker<m1>:LINK:TO:MARKer<m2>` on page 629

`CALCulate<n>:DELTamarker<m>:LINK` on page 629

Assigning the Marker to a Trace
The "Trace" setting assigns the selected marker to an active trace. The trace determines which value the marker shows at the marker position. If the marker was previously assigned to a different trace, the marker remains on the previous frequency or time, but indicates the value of the new trace.

The marker can also be assigned to the currently active trace using the "Marker to Trace" softkey.

If a trace is turned off, the assigned markers and marker functions are also deactivated.

SCPI command:
`CALCulate<n>:MARKer<m>:TRACe` on page 632

Select Marker
Opens a dialog box to select and activate or deactivate one or more markers quickly.

SCPI command:
Marker selected via suffix <m> in remote commands.

All Markers Off
Deactivates all markers in one step.

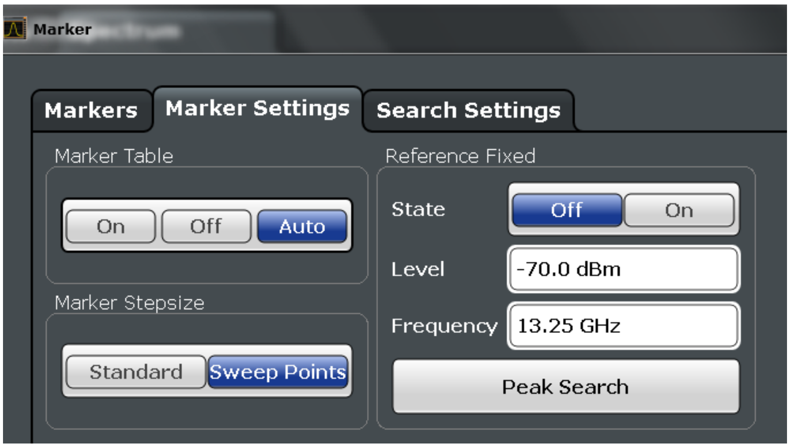
SCPI command:
`CALCulate<n>:MARKer<m>:AOFF` on page 631

General Marker Settings

Some general marker settings allow you to influence the marker behavior for all markers.

These settings are located in the "Marker Settings" tab of the "Marker" dialog box. To display this tab, do one of the following:

- Press the MKR key, then select the "Marker Config" softkey.
- In the "Overview", select "Analysis", and switch to the vertical "Marker" tab. Then select the horizontal "Marker Settings" tab.



Marker Table Display

Defines how the marker information is displayed.

"On" Displays the marker information in a table in a separate area beneath the diagram.

"Off" Displays the marker information within the diagram area.

"Auto" (Default) Up to two markers are displayed in the diagram area. If more markers are active, the marker table is displayed automatically.

SCPI command:

`DISPlay:MTABle` on page 633

Marker Stepsize

Defines the size of the steps that the marker position is moved using the rotary knob.

"Standard" The marker position is moved from pixel to pixel on the display. This is the default and most suitable to move the marker over a larger distance.

"Sweep Points" The marker position is moved from one sweep point to the next. This setting is required for a very precise positioning if more sweep points are collected than the number of pixels that can be displayed on the screen.

SCPI command:

`CALCulate:MARKer:X:SSIZe` on page 633

Defining a Fixed Reference

Instead of using a reference marker that may vary its position depending on the measurement results, a fixed reference marker can be defined for trace analysis.

When you set the "State" to "On", a vertical and a horizontal red display line are displayed, marked as "FXD". The normal marker 1 is activated and set to the peak value of the trace assigned to marker 1, and a delta marker to the next peak. The fixed reference marker is set to the position of marker 1 at the peak value. The delta marker refers to the fixed reference marker.

If activated, the fixed reference marker ("FXD") can also be selected as a "Reference Marker" instead of another marker.

The "Level" and "Frequency" or "Time" settings define the position and value of the reference marker.

Alternatively, a **Peak Search** can be performed to set the current maximum value of the trace assigned to marker 1 as the fixed reference marker.

SCPI command:

`CALCulate<n>:DELTamarker<m>:FUNCtion:FIXed:RPOint:Y` on page 651

`CALCulate<n>:DELTamarker<m>:FUNCtion:FIXed:RPOint:X` on page 651

`CALCulate<n>:DELTamarker<m>:FUNCtion:FIXed:RPOint:MAXimum[:PEAK]` on page 651

6.4.2.2 Marker Search Settings and Positioning Functions

Several functions are available to set the marker to a specific position very quickly and easily, or to use the current marker position to define another characteristic value. In order to determine the required marker position, searches may be performed. The search results can be influenced by special settings.

Most marker positioning functions and the search settings are available in the MKR -> menu.

Search settings are also available via the MARKER key or in the vertical "Marker Config" tab of the "Analysis" dialog box (horizontal "Search Settings" tab).

For more information on searching for signal peaks see chapter 6.4.1.4, "Searching for Signal Peaks", on page 268.

The remote commands required to define these settings are described in chapter 10.6.3.4, "Positioning the Marker", on page 636.

Marker Search Settings

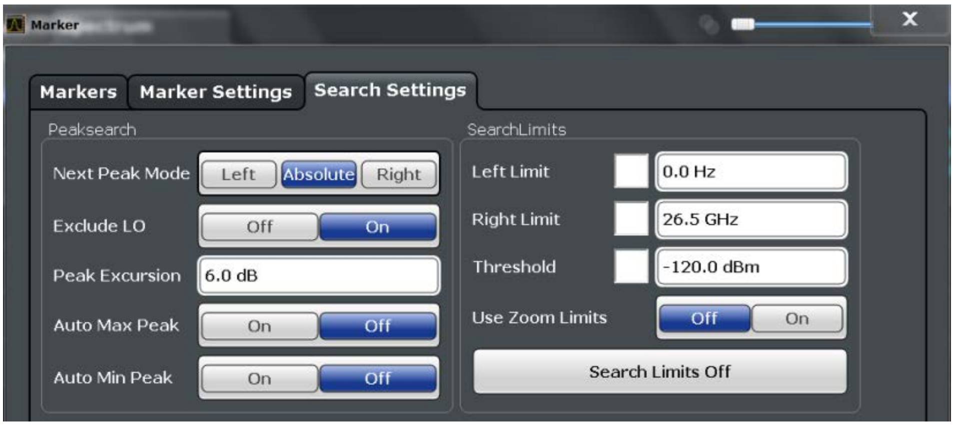
Markers are commonly used to determine peak values, i.e. maximum or minimum values, in the measured signal. Configuration settings allow you to influence the peak search results.

For Spectrograms, special marker settings are available, see "Marker Search Settings for Spectrograms" on page 283.

These settings are are available as softkeys in the "Marker To" menu, or in the "Search Settings" tab of the "Marker" dialog box. To display this tab, do one of the following:

- Press the MKR key, then select the "Marker Config" softkey. Then select the horizontal "Search Settings" tab.

- In the "Overview", select "Analysis", and switch to the vertical "Marker Config" tab. Then select the horizontal "Search Settings" tab.



Search Mode for Next Peak

Selects the search mode for the next peak search.

"Left"	Determines the next maximum/minimum to the left of the current peak.
"Absolute"	Determines the next maximum/minimum to either side of the current peak.
"Right"	Determines the next maximum/minimum to the right of the current peak.

SCPI command:

CALCulate<n>:DELTamarker<m>:MAXimum:LEFT on page 639
CALCulate<n>:MARKer<m>:MAXimum:LEFT on page 637
CALCulate<n>:DELTamarker<m>:MAXimum:NEXT on page 640
CALCulate<n>:MARKer<m>:MAXimum:NEXT on page 637
CALCulate<n>:DELTamarker<m>:MAXimum:RIGHt on page 640
CALCulate<n>:MARKer<m>:MAXimum:RIGHt on page 638
CALCulate<n>:DELTamarker<m>:MINimum:LEFT on page 640
CALCulate<n>:MARKer<m>:MINimum:LEFT on page 638
CALCulate<n>:DELTamarker<m>:MINimum:NEXT on page 641
CALCulate<n>:MARKer<m>:MINimum:NEXT on page 639
CALCulate<n>:DELTamarker<m>:MINimum:RIGHt on page 641
CALCulate<n>:MARKer<m>:MINimum:RIGHt on page 639

Exclude LO

If activated, restricts the frequency range for the marker search functions.

"ON"	The minimum frequency included in the peak search range is ≥ 5 × resolution bandwidth (RBW). Due to the interference by the first local oscillator to the first intermediate frequency at the input mixer, the LO is represented as a signal at 0 Hz. To avoid the peak marker jumping to the LO signal at 0 Hz, this frequency is excluded from the peak search.
"OFF"	No restriction to the search range. The frequency 0 Hz is included in the marker search functions.

SCPI command:
CALCulate:MARKer:LOEXclude on page 634

Peak Excursion

Defines the minimum level value by which a signal must rise or fall so that it will be identified as a maximum or a minimum by the search functions.

Entries from 0 dB to 80 dB are allowed; the resolution is 0.1 dB. The default setting for the peak excursion is 6 dB.

For more information see chapter 6.4.1.4, "Searching for Signal Peaks", on page 268.

SCPI command:
CALCulate<n>:MARKer:PEXCursion on page 634

Automatic Peak Search

If activated, a peak search is performed automatically for marker 1 after each sweep.

For spectrogram displays, define which frame the peak is to be searched in.

SCPI command:
CALCulate<n>:MARKer<m>:MAXimum:AUTO on page 637
CALCulate<n>:MARKer<m>:MINimum:AUTO on page 638

Search Limits

The search results can be restricted by limiting the search area or adding search conditions.

Search Limits (Left / Right) ← Search Limits

If activated, limit lines are defined and displayed for the search. Only results within the limited search range are considered.

For details on limit lines for searches see "Peak search limits" on page 269.

SCPI command:
CALCulate:MARKer:X:SLIMits[:STATe] on page 634
CALCulate:MARKer:X:SLIMits:LEFT on page 635
CALCulate:MARKer:X:SLIMits:RIGHT on page 635

Search Threshold ← Search Limits

Defines an absolute threshold as an additional condition for the peak search. Only peaks that exceed the threshold are detected.

SCPI command:
CALCulate:THReshold on page 636

Using Zoom Limits ← Search Limits
If activated, the peak search is restricted to the active zoom area defined for a single zoom (see "Single Zoom" on page 233).

SCPI command:
`CALCulate:MARKer:X:SLIMits:ZOOM[:STATe]` on page 636

Deactivating All Search Limits ← Search Limits
Deactivates the search range limits.

SCPI command:
`CALCulate:MARKer:X:SLIMits[:STATe]` on page 634
`CALCulate:THReshold:STATe` on page 636

Marker Search Settings for Spectrograms

Spectrograms show not only the current sweep results, but also the sweep history. Thus, when searching for peaks, you must define the search settings within a single time frame (x-direction) and within several time frames (y-direction).

These settings are only available for spectrogram displays.

These settings are are available in the "Search Settings" tab of the "Marker" dialog box. To display this tab, do one of the following:

- Press the MKR key, then select the "Marker Config" softkey. Then select the horizontal "Search Settings" tab.
- In the "Overview", select "Analysis", and switch to the vertical "Marker Config" tab. Then select the horizontal "Search Settings" tab.

Search Mode for Next Peak in X Direction

Selects the search mode for the next peak search within the currently selected frame.

"Left" Determines the next maximum/minimum to the left of the current peak.

"Absolute" Determines the next maximum/minimum to either side of the current peak.

"Right" Determines the next maximum/minimum to the right of the current peak.

SCPI command:

CALCulate<n>:MARKer<m>:MAXimum:LEFT on page 637
CALCulate<n>:MARKer<m>:MAXimum:NEXT on page 637
CALCulate<n>:MARKer<m>:MAXimum:RIGHt on page 638
CALCulate<n>:MARKer<m>:MINimum:LEFT on page 638
CALCulate<n>:MARKer<m>:MINimum:NEXT on page 639
CALCulate<n>:MARKer<m>:MINimum:RIGHt on page 639

Search Mode for Next Peak in Y Direction

Selects the search mode for the next peak search within all frames at the current marker position.

"Up" Determines the next maximum/minimum above the current peak (in more recent frames).

"Absolute" Determines the next maximum/minimum above or below the current peak (in all frames).

"Down" Determines the next maximum/minimum below the current peak (in older frames).

SCPI command:

CALCulate:MARKer<m>:SGRam:Y:MAXimum:ABOVe on page 645
CALCulate:DELTamarker<m>:SGRam:Y:MAXimum:ABOVe on page 649
CALCulate:MARKer<m>:SGRam:Y:MAXimum:BELow on page 646
CALCulate:DELTamarker<m>:SGRam:Y:MAXimum:BELow on page 649
CALCulate:MARKer<m>:SGRam:Y:MAXimum:NEXT on page 646
CALCulate:DELTamarker<m>:SGRam:Y:MAXimum:NEXT on page 649
CALCulate:MARKer<m>:SGRam:Y:MINimum:ABOVe on page 646
CALCulate:DELTamarker<m>:SGRam:Y:MINimum:ABOVe on page 650
CALCulate:MARKer<m>:SGRam:Y:MINimum:BELow on page 646
CALCulate:DELTamarker<m>:SGRam:Y:MINimum:BELow on page 650
CALCulate:MARKer<m>:SGRam:Y:MINimum:NEXT on page 647
CALCulate:DELTamarker<m>:SGRam:Y:MINimum:NEXT on page 650

Marker Search Type

Defines the type of search to be performed in the spectrogram.

"X-Search" Searches only within the currently selected frame.

"Y-Search" Searches within all frames but only at the current marker position.

"XY-Search" Searches in all frames at all positions.

SCPI command:

CALCulate:MARKer<m>:SGRam:XY:MAXimum[:PEAK] on page 645
CALCulate:DELTamarker<m>:SGRam:XY:MAXimum[:PEAK] on page 648
CALCulate:MARKer<m>:SGRam:XY:MINimum[:PEAK] on page 645
CALCulate:DELTamarker<m>:SGRam:XY:MINimum[:PEAK] on page 649
CALCulate:MARKer<m>:SGRam:Y:MAXimum[:PEAK] on page 646
CALCulate:DELTamarker<m>:SGRam:Y:MAXimum[:PEAK] on page 649
CALCulate:MARKer<m>:SGRam:Y:MINimum[:PEAK] on page 647
CALCulate:DELTamarker<m>:SGRam:Y:MINimum[:PEAK] on page 650
CALCulate<n>:MARKer<m>:MAXimum[:PEAK] on page 638
CALCulate<n>:DELTamarker<m>:MAXimum[:PEAK] on page 640
CALCulate<n>:MARKer<m>:MINimum[:PEAK] on page 639
CALCulate<n>:DELTamarker<m>:MINimum[:PEAK] on page 641

Marker Search Area

Defines which frames the search is performed in.

"Visible" Only the visible frames are searched.

"Memory" All frames stored in the memory are searched.

SCPI command:

CALCulate:MARKer:SGRam:SARea on page 645
CALCulate:DELTamarker<m>:SGRam:SARea on page 648

Exclude LO

If activated, restricts the frequency range for the marker search functions.

"ON" The minimum frequency included in the peak search range is ≥ 5 × resolution bandwidth (RBW).
Due to the interference by the first local oscillator to the first intermediate frequency at the input mixer, the LO is represented as a signal at 0 Hz. To avoid the peak marker jumping to the LO signal at 0 Hz, this frequency is excluded from the peak search.

"OFF" No restriction to the search range. The frequency 0 Hz is included in the marker search functions.

SCPI command:

CALCulate:MARKer:LOEXclude on page 634

Peak Excursion

Defines the minimum level value by which a signal must rise or fall so that it will be identified as a maximum or a minimum by the search functions.

Entries from 0 dB to 80 dB are allowed; the resolution is 0.1 dB. The default setting for the peak excursion is 6 dB.

For more information see chapter 6.4.1.4, "Searching for Signal Peaks", on page 268.

SCPI command:
CALCulate<n>:MARKer:PEXCursion on page 634

Automatic Peak Search
If activated, a peak search is performed automatically for marker 1 after each sweep.

For spectrogram displays, define which frame the peak is to be searched in.

SCPI command:
CALCulate<n>:MARKer<m>:MAXimum:AUTO on page 637
CALCulate<n>:MARKer<m>:MINimum:AUTO on page 638

Search Limits
The search results can be restricted by limiting the search area or adding search conditions.

Search Limits (Left / Right) ← Search Limits
If activated, limit lines are defined and displayed for the search. Only results within the limited search range are considered.

For details on limit lines for searches see "Peak search limits" on page 269.

SCPI command:
CALCulate:MARKer:X:SLIMits[:STATe] on page 634
CALCulate:MARKer:X:SLIMits:LEFT on page 635
CALCulate:MARKer:X:SLIMits:RIGHT on page 635

Search Threshold ← Search Limits
Defines an absolute threshold as an additional condition for the peak search. Only peaks that exceed the threshold are detected.

SCPI command:
CALCulate:THReshold on page 636

Using Zoom Limits ← Search Limits
If activated, the peak search is restricted to the active zoom area defined for a single zoom (see "Single Zoom" on page 233).

SCPI command:
CALCulate:MARKer:X:SLIMits:ZOOM[:STATe] on page 636

Deactivating All Search Limits ← Search Limits
Deactivates the search range limits.

SCPI command:
CALCulate:MARKer:X:SLIMits[:STATe] on page 634
CALCulate:THReshold:STATe on page 636

Positioning Functions

The following functions set the currently selected marker to the result of a peak search or set other characteristic values to the current marker value. These functions are available as softkeys in the "Marker To" menu, which is displayed when you press the MKR -> key.

Peak Search
Sets the selected marker/delta marker to the maximum of the trace. If no marker is active, marker 1 is activated.

For spectrogram displays, define which frame the peak is to be searched in.

SCPI command:

`CALCulate<n>:MARKer<m>:MAXimum[:PEAK]` on page 638

`CALCulate<n>:DELTamarker<m>:MAXimum[:PEAK]` on page 640

Search Next Peak
Sets the selected marker/delta marker to the next (lower) maximum of the assigned trace. If no marker is active, marker 1 is activated.

For spectrogram displays, define which frame the next peak is to be searched in.

SCPI command:

`CALCulate<n>:MARKer<m>:MAXimum:NEXT` on page 637

`CALCulate<n>:DELTamarker<m>:MAXimum:NEXT` on page 640

Search Minimum
Sets the selected marker/delta marker to the minimum of the trace. If no marker is active, marker 1 is activated.

For spectrogram displays, define which frame the minimum is to be searched in.

SCPI command:

`CALCulate<n>:MARKer<m>:MINimum[:PEAK]` on page 639

`CALCulate<n>:DELTamarker<m>:MINimum[:PEAK]` on page 641

Search Next Minimum
Sets the selected marker/delta marker to the next (higher) minimum of the selected trace. If no marker is active, marker 1 is activated.

For spectrogram displays, define which frame the next minimum is to be searched in.

SCPI command:

`CALCulate<n>:MARKer<m>:MINimum:NEXT` on page 639

`CALCulate<n>:DELTamarker<m>:MINimum:NEXT` on page 641

Center Frequency = Marker Frequency
Sets the center frequency to the selected marker or delta marker frequency. A peak can thus be set as center frequency, for example to analyze it in detail with a smaller span.

This function is not available for zero span measurements.

SCPI command:

CALCulate<n>:MARKer<m>:FUNCtion:CENTer on page 563

Reference Level = Marker Level

Sets the reference level to the selected marker level.

SCPI command:

CALCulate<n>:MARKer<m>:FUNCtion:REFerence on page 575

6.4.2.3 Marker Function Configuration

Special marker functions can be selected via the "Marker Function" dialog box.

To display this dialog box, do one of the following:

- Press the MKR FUNC key, then select the "Select Marker Function" softkey.
- In the "Overview", select "Analysis", and switch to the vertical "Marker Function Config" tab.

The remote commands required to define these settings are described in chapter 10.6.3, "Working with Markers", on page 628.

Precise Frequency (Signal Count) Marker

A special marker can be used to determine a particular frequency or time in a measured signal very accurately. Signal counters are configured in the "Signal Count Config" dialog box.

To display the "Signal Count Config" dialog box, do one of the following:

- Press the MKR FUNC key, then select the "Select Marker Function" softkey. Then select the "Signal Count" button. Select the "Signal Count Config" softkey.
- In the "Overview", select "Analysis", and switch to the vertical "Marker Function Config" tab. Then select the "Signal Count" button. Select the "Signal Count Config" softkey.

For details see "Performing a Highly Accurate Frequency Measurement (Signal Count)" on page 270

Signal Count Marker State

Activates or deactivates the special signal count marker function.

When activated, the sweep stops at the reference marker until the signal counter has delivered a result.

SCPI command:

`CALCulate<n>:MARKer<m>:COUNt` on page 662

`CALCulate<n>:MARKer<m>:COUNt:FREQuency?` on page 662

Resolution

Defines the resolution with which the signal is analyzed around the reference marker 1.

SCPI command:

`CALCulate<n>:MARKer<m>:COUNt:RESolution` on page 663

Noise Measurement Marker

For each of the 16 markers noise measurement can be activated. Noise measurement markers are configured in the "Noise Measurement Config" dialog box, using the "Noise Measurement" function.

The individual marker settings correspond to those defined in the "Marker" dialog box (see "Individual Marker Setup" on page 275). Any settings to the marker state or type changed in the "Marker Function" dialog box are also changed in the "Marker" dialog box and vice versa.

To display the "Noise Measurement Config" dialog box, do one of the following:

- Press the MKR FUNC key, then select the "Select Marker Function" softkey. Then select the "Noise Measurement" button. Select the "Noise Meas Config" softkey.

- In the "Overview", select "Analysis", and switch to the vertical "Marker Function Config" tab. Then select the "Noise Measurement" button. Select the "Noise Meas Config" softkey.

For details see "Measuring Noise Density" on page 271.

Marker State

Activates or deactivates the marker in the diagram.

SCPI command:

`CALCulate<n>:MARKer<m>[:STATe]` on page 631

`CALCulate<n>:DELTamarker<m>[:STATe]` on page 630

Marker Type

Toggles the marker type.

The type for marker 1 is always "Normal", the type for delta marker 1 is always "Delta". These types cannot be changed.

Note: If normal marker 1 is the active marker, switching the "Mkr Type" activates an additional delta marker 1. For any other marker, switching the marker type does not activate an additional marker, it only switches the type of the selected marker.

"Normal"	A normal marker indicates the absolute value at the defined position in the diagram.

"Delta" A delta marker defines the value of the marker relative to the specified reference marker (marker 1 by default).

SCPI command:
CALCulate<n>:MARKer<m>[:STATe] on page 631
CALCulate<n>:DELTamarker<m>[:STATe] on page 630

Noise Measurement State

Activates or deactivates noise measurement for the marker in the diagram.

This function is only available for normal markers.

If activated, the marker displays the noise power density measured at the position of the marker.

For details see "Measuring Noise Density" on page 271.

SCPI command:
CALCulate<n>:MARKer<m>:FUNCtion:NOISe[:STATe] on page 655
CALCulate<n>:MARKer<m>:FUNCtion:NOISe:RESult? on page 655

Switching All Noise Measurements Off

Deactivates noise measurement for all markers.

SCPI command:
CALCulate<n>:MARKer<m>:FUNCtion:NOISe[:STATe] on page 655

Phase Noise Measurement Marker

For each of the 16 markers phase noise measurement can be activated. Phase noise measurement markers are configured in the "Phase Noise Config" dialog box, using the "Phase Noise" function.

The individual marker settings correspond to those defined in the "Marker" dialog box. Any settings to the marker state or type changed in the "Marker Function" dialog box are also changed in the "Marker" dialog box and vice versa.

To display the "Phase Noise Config" dialog box, do one of the following:

- Press the MKR FUNC key, then select the "Select Marker Function" softkey. Then select the "Phase Noise" button. Select the "Phase Noise Config" softkey.
- In the "Overview", select "Analysis", and switch to the vertical "Marker Function Config" tab. Then select the "Phase Noise" button. Select the "Phase Noise Config" softkey.

For more information see "Measuring Phase Noise" on page 272.

Phase Noise Measurement State

Activates or deactivates phase noise measurement for the reference point in the diagram.

This function is only available for delta markers.

If activated, the delta markers display the phase noise measured at defined offsets from the reference position.

SCPI command:

`CALCulate<n>:DELTamarker<m>:FUNCtion:PNOise[:STATe]` on page 656

`CALCulate<n>:DELTamarker<m>:FUNCtion:PNOise:RESult?` on page 656

Defining a Reference Point

Instead of using marker 1 as the reference marker, a fixed reference marker can be defined for phase noise measurement.

The "Level" and "Frequency" or "Time" settings define the position and value of the reference point.

Alternatively, a **Peak Search** can be performed to set the maximum value of the selected trace as the reference point.

If "Automatic Peak Search" is activated, a peak search is started automatically after each sweep and the result is used as the reference point.

SCPI command:

`CALCulate<n>:DELTamarker<m>:FUNCtion:FIXed:RPOint:Y` on page 651
`CALCulate<n>:DELTamarker<m>:FUNCtion:FIXed:RPOint:X` on page 651
`CALCulate<n>:DELTamarker<m>:FUNCtion:FIXed:RPOint:MAXimum[:PEAK]` on page 651
`CALCulate<n>:DELTamarker<m>:FUNCtion:PNOise:AUTO` on page 656

Switching All Phase Noise Measurements Off

Deactivates phase noise measurement for all markers.

SCPI command:

`CALCulate<n>:DELTamarker<m>:FUNCtion:PNOise[:STATe]` on page 656

n dB Down Marker

A special marker can be defined to determine a characteristic bandwidth or time span in a measured signal. n dB down markers are configured in the "N dB Down Config" dialog box, using the "n dB down" function.

To display the "N dB Down Config" dialog box, do one of the following:

- Press the MKR FUNC key, then select the "Select Marker Function" softkey. Then select the "n dB down" button. Select the "N dB Down Config" softkey.
- In the "Overview", select "Analysis", and switch to the vertical "Marker Function Config" tab. Then select the "n dB down" button. Select the "N dB Down Config" softkey.

For details see "Measuring Characteristic Bandwidths (n dB Down Marker)" on page 273

n dB down Marker State

Activates or deactivates the special n dB down marker function.

SCPI command:

`CALCulate<n>:MARKer<m>:FUNCtion:NDBDown:STATe` on page 661
`CALCulate<n>:MARKer<m>:FUNCtion:NDBDown:RESult?` on page 660

n dB down Delta Value

Defines the delta level from the reference marker 1 used to determine the bandwidth or time span.

SCPI command:

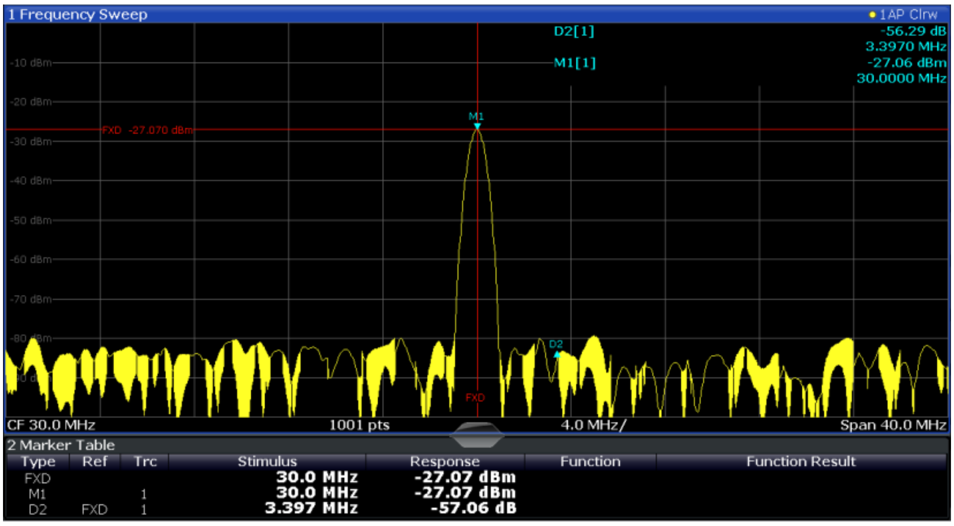
`CALCulate<n>:MARKer<m>:FUNCtion:NDBDown:FREQuency?` on page 660

`CALCulate<n>:MARKer<m>:FUNCtion:NDBDown:TIME` on page 661

Reference Fixed

Instead of using a reference marker that may vary its position depending on the measurement results, a fixed reference marker can be defined for trace analysis. Once positioned, the reference marker does not move during subsequent sweeps unless you explicitely move it manually.

When you select this marker function, a vertical and a horizontal red display line are displayed, marked as "FXD". A normal marker is activated and set to the peak value and a delta marker to the next peak. The fixed reference marker is set to the position of the normal marker at the peak value. The delta marker refers to the fixed reference marker.



You can move the position of the fixed reference marker graphically by dragging the display lines, or numerically by entering values for the marker position and level.

For more information see chapter 6.4.3.2, "How to Use a Fixed Reference Marker", on page 300.

Band Power Marker

For each of the 16 markers band power measurement can be activated. Band power measurement markers are configured in the "Band Power Config" dialog box, using the "Band Power" function.

The individual marker settings correspond to those defined in the "Marker" dialog box (see "Individual Marker Setup" on page 275). Any settings to the marker state or type changed in the "Marker Function" dialog box are also changed in the "Marker" dialog box and vice versa.

To display the "Band Power Config" dialog box, do one of the following:

- Press the MKR FUNC key, then select the "Select Marker Function" softkey. Then select the "Band Power" button. Select the "Band Power Config" softkey.
- In the "Overview", select "Analysis", and switch to the vertical "Marker Function Config" tab. Then select the "Band Power" button. Select the "Band Power Config" softkey.

For more information see "Measuring the Power in a Channel (Band)" on page 274.

Band Power Measurement State

Activates or deactivates band power measurement for the marker in the diagram.

Band power markers are only available for standard frequency measurements (not zero span) in the Spectrum application.

If activated, the markers display the power or density measured in the band around the current marker position.

For details see "Measuring the Power in a Channel (Band)" on page 274.

SCPI command:

`CALCulate<n>:MARKer<m>:FUNCtion:BPOWer[:STATe]` on page 658

Span

Defines the span (band) around the marker for which the power is measured. The span is indicated by lines in the diagram.

SCPI command:

CALCulate<n>:MARKer<m>:FUNCtion:BPOWer:SPAN on page 658

Power Mode

Defines the mode of the power measurement result.

"Power" The result is an absolute power level displayed in dBm.

"Density" The result is a power level in relation to the bandwidth, displayed in dBm/Hz.

SCPI command:

CALCulate<n>:MARKer<m>:FUNCtion:BPOWer:MODE on page 657

Switching All Band Power Measurements Off

Deactivates band power measurement for all markers.

SCPI command:

CALCulate<n>:MARKer<m>:FUNCtion:BPOWer[:STATe] on page 658

Marker Demodulation

A special marker can be used to demodulate the signal at a particular position and send the result to the audio output. Marker Demodulation is configured in the "Marker Demod Config" dialog box, using the "Marker Demodulation" function.

To display the "Marker Demod Config" dialog box, do one of the following:

- Press the MKR FUNC key, then select the "Select Marker Function" softkey. Then select the "Marker Demodulation" button. Select the "Marker Demod Config" softkey.
- In the "Overview", select "Analysis", and switch to the vertical "Marker Function Config" tab. Then select the "Marker Demodulation" button. Select the "Marker Demod Config" softkey.

This function is not available for Spectrum Emission Mask measurements.

For details see "Demodulating Marker Values and Providing Audio Output" on page 274.

Marker Demodulation State

Activates or deactivates the demodulation output.

SCPI command:

`CALCulate<n>:MARKer<m>:FUNCtion:DEModulation[:STATe]` on page 664

Continuous Demodulation

If activated, the signal is demodulated continuously (not only at the marker position) and sent to the audio output. This allows you to monitor the frequency range acoustically (assuming the sweep time is long enough).

For zero span measurements, demodulation is always active continuously.

SCPI command:

`CALCulate<n>:MARKer<m>:FUNCtion:DEModulation:CONTinuous` on page 663

Marker Stop Time

Defines how long the sweep is stopped at the marker position to output the demodulated signal.

For zero span measurements, demodulation is always active continuously, regardless of the marker stop time.

SCPI command:

`CALCulate<n>:MARKer<m>:FUNCtion:DEModulation:HOLDoff` on page 664

Modulation

Defines the demodulation mode for output (AM/FM). The default setting is AM.

SCPI command:

`CALCulate<n>:MARKer<m>:FUNCtion:DEModulation:SELect` on page 664

Squelch

Activates or deactivates the squelch function. If activated, the audible AF is cut off below a defined threshold level. Thus, you avoid hearing noise at the audio output when no signal is available.

The squelch function activates the video trigger function (see "Video" on page 217) and deactivates any other trigger or gating settings. The squelch level and trigger level are set to the same value.

The trigger source in the channel bar is indicated as "SQL" for squelch. The squelch level is indicated by a red line in the diagram.

SCPI command:

`[SENSe:]DEMod:SQUelch[:STATe]` on page 665

Squelch level

Defines the level threshold below which the audible AF is cut off if squelching is enabled. The video trigger level is set to the same value.

The squelch level is indicated by a red line in the diagram.

SCPI command:

`[SENSe:]DEMod:SQUelch:LEVel` on page 665

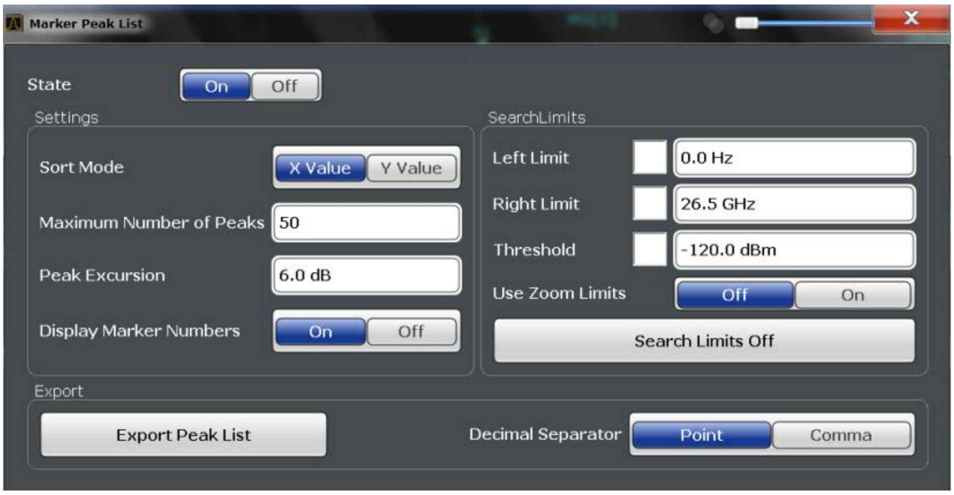
Marker Peak List Configuration

The marker peak list provides an overview of all marker peaks in the measurement. You can define search and sort criteria to influence the results of the analysis. The general marker search settings also apply to the marker peak list (see "Marker Search Settings" on page 280).

For more information see chapter 6.4.1.4, "Searching for Signal Peaks", on page 268.

To display the "Marker Peak List" dialog, do one of the following:

- Press the MKR FUNC key, then select the "Marker Peak List" softkey.
- In the "Overview", select "Analysis", and switch to the vertical "Peak List" tab.



Peak List State

Activates/deactivates the marker peak list. If activated, the peak list is displayed and the peaks are indicated in the trace display.

For each listed peak the frequency/time ("Stimulus") and level ("Response") values are given.

SCPI command:

`CALCulate<n>:MARKer<m>:FUNCtion:FPEaks:STAT` on page 654

Sort Mode
Defines whether the peak list is sorted according to the x-values or y-values. In either case the values are sorted in ascending order.

SCPI command:
`CALCulate<n>:MARKer<m>:FUNCtion:FPEaks:SORT` on page 653

Maximum Number of Peaks
Defines the maximum number of peaks to be determined and displayed.

SCPI command:
`CALCulate<n>:MARKer<m>:FUNCtion:FPEaks:LIST:SIZE` on page 653

Peak Excursion
Defines the minimum level value by which a signal must rise or fall so that it will be identified as a maximum or a minimum by the search functions.

Entries from 0 dB to 80 dB are allowed; the resolution is 0.1 dB. The default setting for the peak excursion is 6 dB.

For more information see chapter 6.4.1.4, "Searching for Signal Peaks", on page 268.

SCPI command:
`CALCulate<n>:MARKer:PEXCursion` on page 634

Displaying Marker Numbers
By default, the marker numbers are indicated in the diagram so you can find the peaks from the list. However, for large numbers of peaks the marker numbers may decrease readability; in this case, deactivate the marker number display.

SCPI command:
`CALCulate<n>:MARKer<m>:FUNCtion:FPEaks:ANNotation:LABel[:STATe]` on page 652

Exporting the Peak List
The peak list can be exported to an ASCII file (`.DAT`) for analysis in an external application.

SCPI command:
`MMEMory:STORe:LIST` on page 696
`FORMat:DEXPort:DSEParator` on page 678

Deactivating All Marker Functions

All special marker functions can be deactivated in one step.

Use the "All Functions Off" button in the "Marker Functions" dialog box.

6.4.3 How to Work With Markers

The following step-by-step instructions demonstrate in detail how to work with markers.

6.4.3.1 How to Analyze a Signal Point in Detail

Step-by-step instructions on working with markers are provided here. For details on individual functions and settings see chapter 6.4.2.1, "Marker Settings", on page 275.

The remote commands required to perform these tasks are described in chapter 10.6.3, "Working with Markers", on page 628.

When you need to analyze a characteristic point in the signal in more detail, the following procedure can be helpful:

1. Perform a peak search to determine the characteristic point roughly by pressing the PEAK SEARCH key.
2. If the required signal point is not the maximum, continue the peak search to one of the subsequent maxima or minima:
   a) Press the MKR -> key.
   b) Select the "Next Peak" or "Next Min" key.
   c) If necessary, change the search settings by selecting the "Search Config" softkey.
3. Center the display around the determined signal point by setting the marker value to the center frequency. Select the "Center = Mkr Freq" softkey.
4. Determine the precise frequency of the signal point:
   a) Select the "Select Marker Function" softkey.
   b) Select the "Signal Count" button.
   c) Select the "Signal Count Resolution" softkey.
   d) Select the resolution depending on how precise the result needs to be.

6.4.3.2 How to Use a Fixed Reference Marker

By default, delta markers refer to marker 1. However, they can also refer to a fixed reference marker.

How to Define and Move a Fixed Reference Marker

1. To display a fixed reference marker, do one of the following:
   - Press the MKR FUNC key, then select the "Reference Fixed" marker function.
   - In the "Marker" dialog box, in the "Reference Fixed" area of the "Marker Config" tab, set the "State" to "On".

A vertical and a horizontal red display line are displayed, marked as "FXD". The normal marker 1 is activated and set to the peak value of the trace assigned to marker 1, and a delta marker to the next peak. The fixed reference marker is set to the position of marker 1 at the peak value.

2. To move the fixed reference marker, do one of the following:
   - Change the "Level" and "Frequency" of the reference point in the "Marker Config" tab of the "Marker" dialog box, . By default, the current peak value of trace 1 is set.
   - Set the fixed reference marker to the current peak value by selecting the "Peak Search" button in the "Marker Config" tab of the "Marker" dialog box.
   - Move the "FXD" display lines that define the position of the fixed reference marker by dragging them on the screen.

How to Assign a Fixed Reference Marker to Delta Markers

1. In the "Marker" dialog box, select the horizontal "Markers" tab.
2. For the active delta marker that is to refer to the fixed reference marker, select "FXD" from the "Ref. Marker" list.

   The delta marker indicates the offset of the current trace value at the marker position from the fixed reference value.

6.4.3.3 How to Output the Demodulated Signal Accoustically

For long sweep times you may wish to monitor a measurement accoustically rather than visually to determine when a certain signal level is reached.

CAUTION

Risk of hearing damage

To protect your hearing, make sure that the volume setting is not too high before putting on the headphones.

1. Set marker 1 to the signal level you want to monitor.
2. Press the MKR FUNCT key.
3. Select the "Select Marker Function" softkey.
4. Select the "Marker Demodulation" button.
5. Select the "Marker Demod Config" softkey.

   The marker function results are determined immediately according to the default settings.
6. Define how long you want to hear the output signal when the marker value is reached by entering the duration in the "Marker Stop Time" field.

Alternatively, the audio signal can be output continuously, regardless of the marker value; in this case, set "Continuous Demodulation" to "On".

7. Select the modulation type (AM/FM/PM) of the signal.
8. To avoid listening to noise during continuous output, set "Squelch" to "On" and define the signal level below which the signal is ignored ("Squelch level").
9. Set "Marker Demodulation" to "On".
10. Plug your headphones into the PHONES connector on the front panel of the R&S FSW.
11. Adjust the volume using the rotary knob next to the PHONES connector.

    During the next or currently running measurement, when the sweep reaches the marker position, the demodulated signal is output as an audio signal via the headphones for the given duration. Or, depending on the configuration, the demodulated signal is continuously output via the headphones, if the signal level exceeds the squelch level.

6.4.4 Measurement Example: Measuring Harmonics Using Marker Functions

This measurement example describes how to measure harmonics using the provided marker functions. Note that this task can be performed much simpler using the Harmonic Distortion measurement (see chapter 4.9, "Harmonic Distortion Measurement", on page 136).

Signal generator settings (e.g. R&S FSW SMU):

Frequency:	128 MHz
Level:	- 25 dBm

Procedure:

1. Preset the R&S FSW.
2. Set the center frequency to *128 MHz*.
3. Set the span to *100 kHz*.

   The R&S FSW displays the reference signal with a span of 100 kHz and resolution bandwidth of 1 kHz.
4. Switch on the marker by pressing the MKR key.
   The marker is positioned on the trace maximum.
5. Set the measured signal frequency and the measured level as reference values:
   a) Press the MKR FUNC key
   b) Press the "Reference Fixed" softkey.

The position of the marker becomes the reference point. The reference point level is indicated by a horizontal line, the reference point frequency with a vertical line. At the same time, the delta marker 2 is switched on.

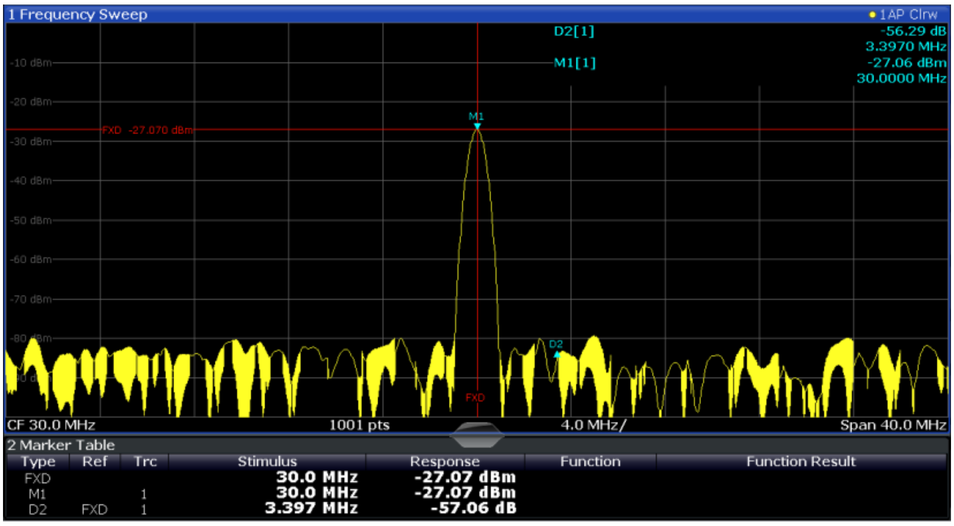


Fig. 6-13: Fundamental wave and the frequency and level reference point

6. Make the step size for the center frequency correspond to the signal frequency: in the "Frequency" configuration dialog box, select "Center Frequency Stepsize = Marker".

   The step size for the center frequency is now equal to the marker frequency.

7. Move the center frequency to the 2nd harmonic of the signal by pressing the UP (⇧) key on the front panel.

   The center frequency is set to the 2nd harmonic.

8. Place the delta marker on the 2nd harmonic: in the "Marker To" menu, select the "Peak" softkey.

   The delta marker moves to the maximum of the 2nd harmonic. The displayed level result is relative to the reference point level (= fundamental wave level).

The other harmonics are measured by repeating steps step 7 and step 8, with the center frequency being incremented or decremented in steps of 128 MHz using the UP or DOWN keys.

6.5 Display and Limit Lines

Display and limit lines help you analyze a measurement trace.

6.5.1 Basics on Display Lines

Display lines help you analyze a trace – as do markers. The function of a display line is comparable to that of a ruler that can be shifted on the trace in order to mark absolute values. They are used exclusively to visually mark relevant frequencies or points in time (zero span), as well as constant level values. It is not possible to check automatically whether the points are below or above the marked level values - use limit lines for that task (see chapter 6.5.2, "Basics on Limit Lines", on page 304).

Two different types of display lines are provided:

- Two horizontal level lines for marking levels – Display Line 1 and 2
  The level lines are continuous horizontal lines across the entire width of a diagram and can be shifted in y direction.
- Two vertical frequency or time lines for marking frequencies or points in time – Frequency/Time Line 1 and 2
  The frequency or time lines are continuous vertical lines across the entire height of the diagram and can be shifted in x direction.

Lables

Each line is identified by one of the following abbreviations in the diagrams:

- D1: Display Line 1
- D2: Display Line 2
- F1: Frequency Line 1
- F2: Frequency Line 2
- T1: Time Line 1
- T2: Time Line 2

6.5.2 Basics on Limit Lines

Limit lines are used to define amplitude curves or spectral distribution boundaries in the result diagram which are not to be exceeded. They indicate, for example, the upper limits for interference radiation or spurious waves which are allowed from a device under test (DUT). When transmitting information in TDMA systems (e.g. GSM), the amplitude of the bursts in a time slot must adhere to a curve that falls within a specified tolerance band. The lower and upper limits may each be specified by a limit line. Then, the amplitude curve can be controlled either visually or automatically for any violations of the upper or lower limits (GO/NOGO test).

The R&S FSW supports limit lines with a maximum of 200 data points. Eight of the limit lines stored in the instrument can be activated simultaneously. The number of limit lines stored in the instrument is only limited by the capacity of the storage device used.

Compatibility

Limit lines are compatible with the current measurement settings, if the following applies:

- The x unit of the limit line has to be identical to the current setting.

- The y unit of the limit line has to be identical to the current setting with the exception of dB based units; all dB based units are compatible with each other.

Validity

Only limit lines that fulfill the following conditions can be activated:

- Each limit line must consist of a minimum of 2 and a maximum of 200 data points.
- The frequencies/times for each data point must be defined in ascending order; however, for any single frequency or time, two data points may be entered (to define a vertical segment of a limit line).
- Gaps in frequency or time are not allowed. If gaps are desired, two separate limit lines must be defined and then both enabled.
- The entered frequencies or times need not necessarily be selectable in R&S FSW. A limit line may also exceed the specified frequency or time range. The minimum frequency for a data point is -200 GHz, the maximum frequency is 200 GHz. For the time range representation, negative times may also be entered. The allowed range is -1000 s to +1000 s.

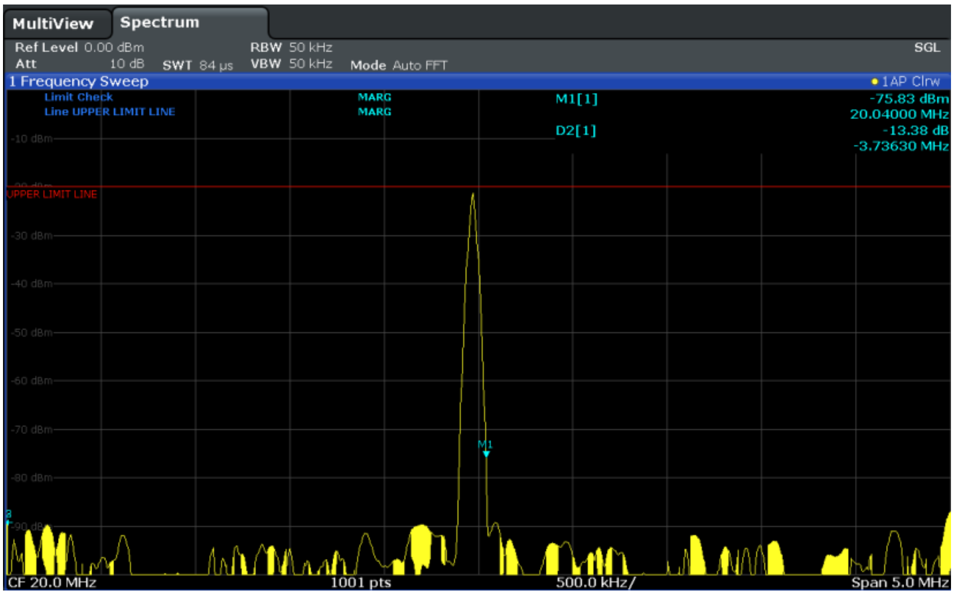


Fig. 6-14: Example for an upper limit line

Limits and Margins

Limit lines define strict values that must not be exceeded by the measured signal. A **margin** is similar to a limit, but less strict and it still belongs to the valid data range. It can be used as a warning that the limit is almost reached. The margin is not indicated by a separate line in the display, but if it is violated, a warning is displayed. Margins are defined as lines with a fixed distance to the limit line.

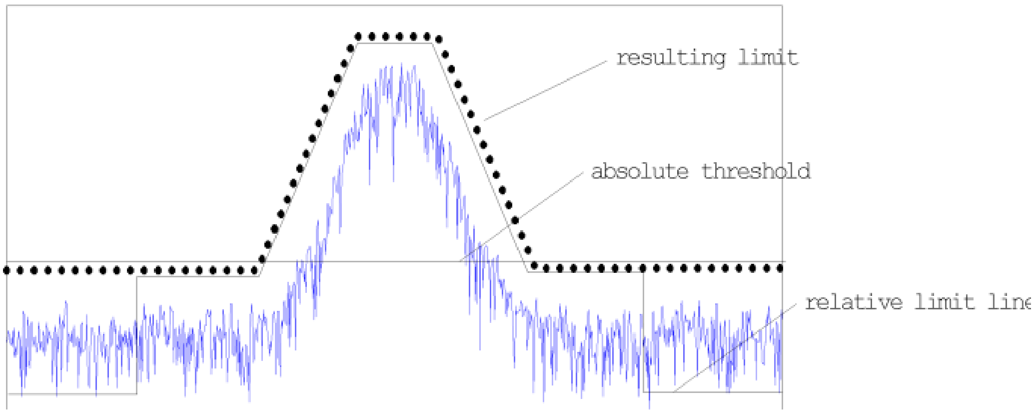
To check the signal for maximum levels you must define an **upper limit**, whereas to check the signal for minimum levels you must define a **lower limit**.

Limits can be defined relative to the reference level, the beginning of the time scale, or the center frequency, or as absolute values.

Relative scaling is suitable, for example, if masks for bursts are to be defined in zero span, or if masks for modulated signals are required in the frequency domain.

Thresholds

If the y-axis for the limit line data points uses relative scaling, an additional absolute **threshold** can be defined for the limit check. In this case, both the threshold value and the relative limit line must be exceeded before a violation occurs.



Offsets and Shifting

A configured limit line can easily be moved vertically or horizontally. Two different methods to do so are available:

- An **offset** moves the entire line in the diagram without editing the configured values or positions of the individual data points. This option is only available if relative scaling is used.
  Thus, a new limit line can be easily generated based upon an existing limit line which has been shifted horizontally or vertically.
- Defining a **shift** width for the values or position of the individual data points changes the line configuration, thus changing the position of the line in the diagram.

Limit Check Results

A limit check is automatically performed as soon as any of the limit lines is activated ("Visibility" setting). Only the specified "Traces to be Checked" are compared with the active limit lines. The status of the limit check for each limit line is indicated in the diagram. If a violation occurs, the limit check status is set to "MARG" for a margin violation, or to"FAIL" for a limit violation.

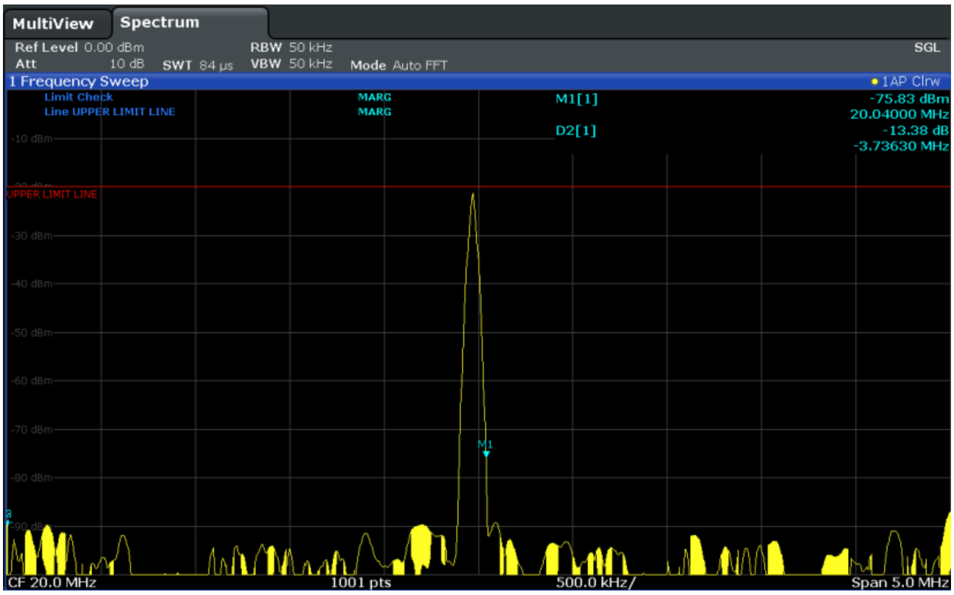


Fig. 6-15: Margin violation for limit check

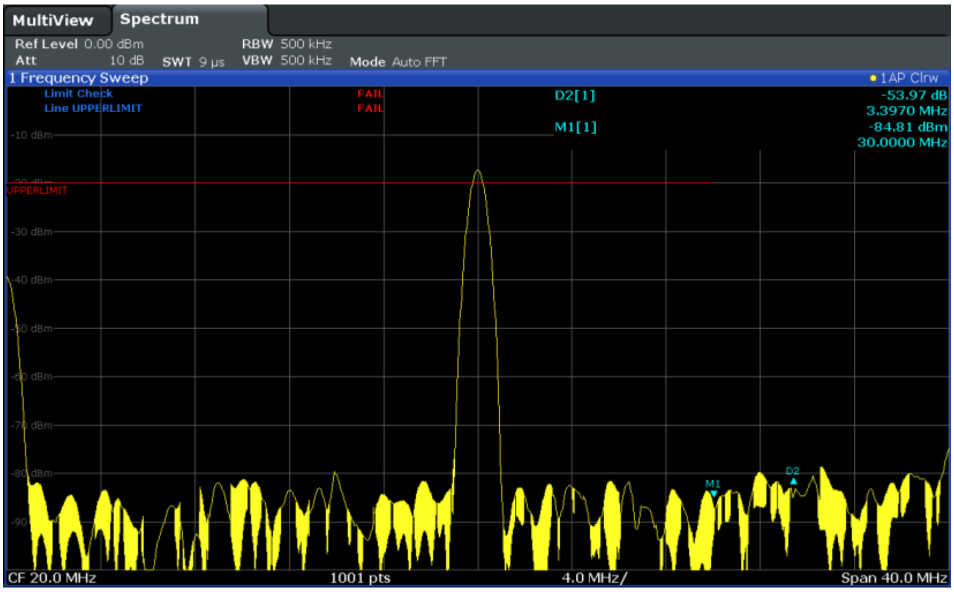


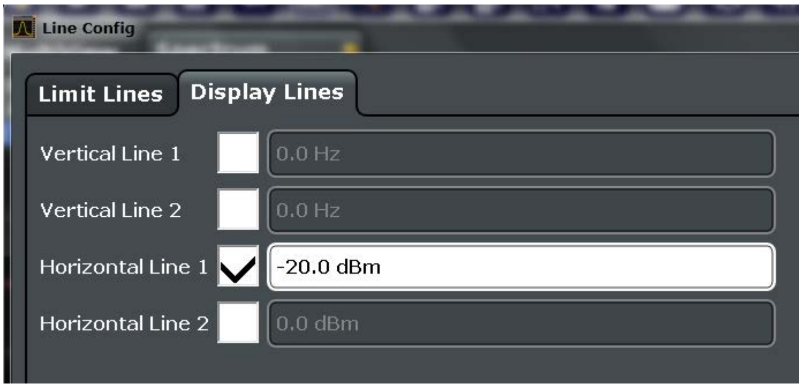
Fig. 6-16: Limit violation for limit check

6.5.3 Settings and Functions for Display and Limit Lines

For remote operation, see chapter 10.6.4, "Configuring Display and Limit Lines", on page 665.

6.5.3.1 Display Line Settings

Two vertical and two horizontal lines can be defined in the display.



Vertical Line 1/2

Activates a vertical display line (F1/F2 or T1/T2) in the diagram at the specified frequency or point in time, depending on the frequency span.

SCPI command:

`CALCulate<n>:FLINe<k>` on page 666

`CALCulate<n>:FLINe<k>` on page 666

`CALCulate<n>:TLINe<Line>` on page 667

`CALCulate<n>:TLINe<Line>` on page 667

Horizontal Line 1/2

Activates a horizontal display line (D1/D2) in the diagram at the specified level.

SCPI command:

`CALCulate<n>:DLINe<k>` on page 666

`CALCulate<n>:DLINe<k>` on page 666

6.5.3.2 Limit Line Settings and Functions

Up to 8 limit lines can be displayed simultaneously in the R&S FSW. Many more can be stored on the instrument.

Limit Line Management

Limit lines are managed in the "Line Config" dialog box which is displayed when you press the LINES key and then "Lines Config" softkey.

For the limit line overview, the R&S FSW searches for all stored limit lines with the file extension `.LIM` in the `limits` subfolder of the main installation folder. The overview allows you to determine which limit lines are available and can be used for the current measurement.

For details on settings for individual lines see "Limit Line Details" on page 311.

For more basic information on limit lines see chapter 6.5.2, "Basics on Limit Lines", on page 304.

Name

The name of the stored limit line.

Unit
The unit in which the y-values of the data points of the limit line are defined.

Compatibility
Indicates whether the limit line definition is compatible with the current measurement settings.

For more information on which conditions a limit line must fulfill to be compatible, see chapter 6.5.2, "Basics on Limit Lines", on page 304.

Visibility
Displays or hides the limit line in the diagram. Up to 8 limit lines can be visible at the same time. Inactive limit lines can also be displayed in the diagram.

SCPI command:
`CALCulate:LIMit<k>:LOWer:STATe` on page 671
`CALCulate:LIMit<k>:UPPer:STATe` on page 674
`CALCulate:LIMit:ACTive?` on page 675

Traces to be Checked
Defines which traces are automatically checked for conformance with the limit lines. As soon as a trace to be checked is defined, the assigned limit line is active. One limit line can be activated for several traces simultaneously. If any of the "Traces to be Checked" violate any of the active limit lines, a message is indicated in the diagram.

SCPI command:
`CALCulate:LIMit<k>:TRACe` on page 675

Comment
An optional description of the limit line.

Included Lines in Overview (View Filter)
Defines which of the stored lines are included in the overview.

"Show compatible"	Only compatible lines Whether a line is compatible or not is indicated in the Compatibility setting.
"Show all"	All stored limit lines with the file extension `.LIM` in the `limits` subfolder of the main installation folder (if not restricted by "Show lines for all modes" setting).

Show lines for all modes
If activated (default), limit lines from all applications are displayed. Otherwise, only lines that were created in the Spectrum application are displayed.

Note that limit lines from some applications may include additional properties that are lost when the limit lines are edited in the Spectrum application. In this case a warning is displayed when you try to store the limit line.

X-Offset
Shifts a limit line that has been specified for relative frequencies or times (x-axis) horizontally.

This setting does not have any effect on limit lines that are defined by absolute values for the x-axis.

SCPI command:

CALCulate:LIMit<k>:CONTrol:OFFSet on page 669

Y-Offset

Shifts a limit line that has relative values for the y-axis (levels or linear units such as volt) vertically.

This setting does not have any effect on limit lines that are defined by absolute values for the y-axis.

SCPI command:

CALCulate:LIMit<k>:LOWer:OFFSet on page 670

CALCulate:LIMit<k>:UPPer:OFFSet on page 673

Create New Line

Creates a new limit line.

Edit Line

Edit an existing limit line configuration.

Copy Line

Copy the selected limit line configuration to create a new line.

SCPI command:

CALCulate:LIMit<k>:COPY on page 675

Delete Line

Delete the selected limit line configuration.

SCPI command:

CALCulate:LIMit<k>:DELete on page 675

Disable All Lines

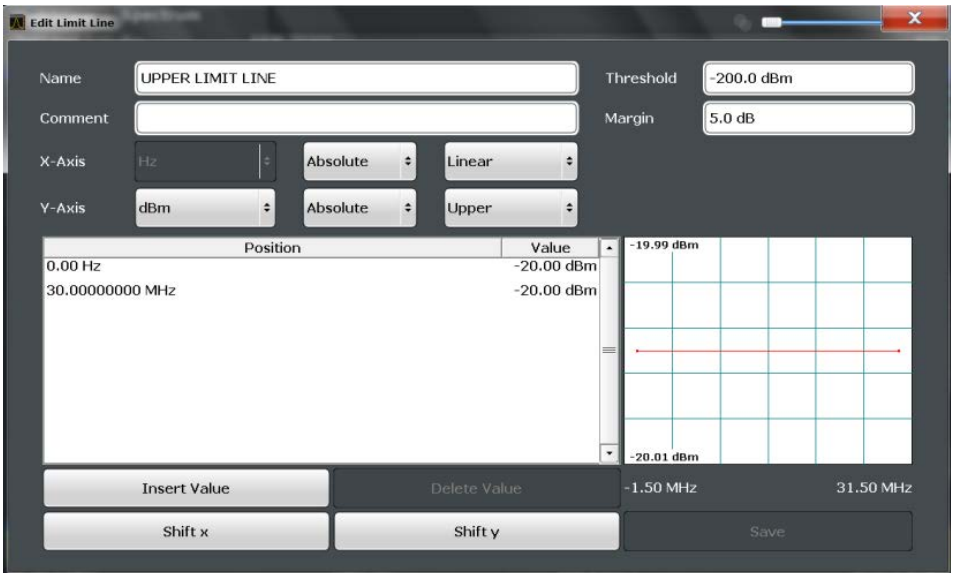
Disable all limit lines in one step.

SCPI command:

CALCulate:LIMit<k>:STATe on page 676

Limit Line Details

Limit lines details are configured in the "Edit Line Line" dialog box which is displayed when you select the "New", "Edit" or "Copy To" buttons in the "Line Config" dialog box.



Name

Defines the limit line name. All names must be compatible with Windows conventions for file names. The limit line data is stored under this name (with a `.LIM` extension).

SCPI command:

`CALCulate:LIMit<k>:NAME` on page 672

Comment

Defines an optional comment for the limit line. The text may contain up to 40 characters.

SCPI command:

`CALCulate:LIMit:COMMent` on page 668

Threshold

Defines an absolute threshold value (only for relative scaling of the y-axis).

For details on thresholds see chapter 6.5.2, "Basics on Limit Lines", on page 304.

SCPI command:

`CALCulate:LIMit<k>:LOWer:THReshold` on page 671

`CALCulate:LIMit<k>:UPPer:THReshold` on page 674

Margin
Defines a margin for the limit line. The default setting is 0 dB (i.e. no margin).

For details on margins see chapter 6.5.2, "Basics on Limit Lines", on page 304.

SCPI command:

`CALCulate:LIMit<k>:LOWer:MARGin` on page 670

`CALCulate:LIMit<k>:UPPer:MARGin` on page 673

X-Axis
Describes the horizontal axis on which the data points of the limit line are defined. Includes the following settings:

- Domain:
  - "Hz": for frequency domain
  - "s": for time domain
- Scaling mode: absolute or relative (Hz/s/%) values
  For relative values, the frequencies are referred to the currently set center frequency. In the zero span mode, the left boundary of the diagram is used as the reference.
- Scaling: linear or logarithmic

SCPI command:

`CALCulate:LIMit<k>:LOWer:SPACing` on page 671

`CALCulate:LIMit<k>:UPPer:SPACing` on page 674

`CALCulate:LIMit<k>:LOWer:MODE` on page 670

`CALCulate:LIMit<k>:UPPer:MODE` on page 673

`CALCulate:LIMit<k>:CONTrol:DOMain` on page 668

Y-Axis
Describes the vertical axis on which the data points of the limit line are defined. Includes the following settings:

- Level unit
- Scaling mode: absolute or relative (dB/%) values
  Relative limit values refer to the reference level.
- Limit type: upper or lower limit; values must stay above the lower limit and below the upper limit to pass the limit check

SCPI command:

`CALCulate:LIMit<k>:UNIT` on page 672

`CALCulate:LIMit<k>:LOWer:SPACing` on page 671

`CALCulate:LIMit<k>:UPPer:SPACing` on page 674

Data points
Each limit line is defined by a minimum of 2 and a maximum of 200 data points. Each data point is defined by its position (x-axis) and value (y-value). Data points must be defined in ascending order. The same position can have two different values.

SCPI command:

`CALCulate:LIMit<k>:CONTrol[:DATA]` on page 668

`CALCulate:LIMit<k>:LOWer[:DATA]` on page 670

`CALCulate:LIMit<k>:UPPer[:DATA]` on page 672

Insert Value

Inserts a data point in the limit line above the selected one in the "Edit Limit Line" dialog box.

Delete Value

Deletes the selected data point in the "Edit Limit Line" dialog box.

Shift x

Shifts the x-value of each data point horizontally by the defined shift width (as opposed to an additive offset defined for the entire limit line, see "X-Offset" on page 310).

SCPI command:

`CALCulate:LIMit<k>:CONTrol:SHIFt` on page 669

Shift y

Shifts the y-value of each data point vertically by the defined shift width (as opposed to an additive offset defined for the entire limit line, see "Y-Offset" on page 311).

SCPI command:

`CALCulate:LIMit<k>:LOWer:SHIFt` on page 671

`CALCulate:LIMit<k>:UPPer:SHIFt` on page 673

Save

Saves the currently edited limit line under the name defined in the "Name" field.

6.5.4 How to Work with Display and Limit Lines

Step-by-step instructions on configuring display and limit lines are provided here. For details on individual functions and settings see chapter 6.5.3, "Settings and Functions for Display and Limit Lines ", on page 307.

The remote commands required to perform these tasks are described in chapter 10.6.4, "Configuring Display and Limit Lines", on page 665.

6.5.4.1 Defining Display Lines

1. Display lines are configured in the "Lines Config" dialog box. To display this dialog box, press the LINES key and then "Lines Config".
2. Select the "Display Lines" tab.
3. To define a vertical line, select "Vertical Line 1" or 2 and enter the x-value at which the line is to be displayed.
   To define a horizontal line, select "Horizontal Line 1" or 2 and enter the y-value at which the line is to be displayed.

6.5.4.2 Defining Limit Lines

Limit lines are configured in the "Lines Config" dialog box. To display this dialog box, do one of the following:

- Press the LINES key and then the "Lines Config" softkey, then select the "Lines Config" tab.
- In the "Overview", select "Analysis" and then the vertical "Limit Lines Config" tab.

Limit lines for spurious and SEM measurements

Note that for spurious and SEM measurements, special limit lines can be defined for each frequency range, see chapter 4.5.4.2, "Limit Lines in SEM Measurements", on page 78 and chapter 4.6.3.2, "Limit Lines in Spurious Measurements", on page 107. It is strongly recommended that you define limits only via the "Sweep list" dialog for these measurements, not using the LINES key.

Any changes to the special limit lines are automatically overwritten when the sweep list settings are changed.

The following tasks are described here:

- "How to find compatible limit lines" on page 315
- "How to activate and deactivate a limit check" on page 315
- "How to edit existing limit lines" on page 316
- "How to copy an existing limit line" on page 316
- "How to delete an existing limit line" on page 316
- "How to configure a new limit line" on page 316
- "How to move the limit line vertically or horizontally" on page 317

How to find compatible limit lines

► In the "Line Config" dialog box, select the "View filter" option: "Show compatible".

All stored limit lines with the file extension `.LIM` in the `limits` subfolder of the main installation folder of the instrument that are compatible to the current measurement settings are displayed in the overview.

How to activate and deactivate a limit check

A limit check is automatically performed as soon as any of the limit lines is activated.

1. To activate a limit check:
   Select the "Check Traces" setting for a limit line in the overview and select the trace numbers to be included in the limit check. One limit line can be assigned to several traces.

   The specified traces to be checked are compared with the active limit lines. The status of the limit check is indicated in the diagram.

2. To deactivate a limit line, deactivate all "Traces to check" for it.
   To deactivate all limit lines at once, select the "Disable All Lines" button.

The limit checks for the deactivated limit lines are stopped and the results are removed form the display.

How to edit existing limit lines

Existing limit line configurations can be edited.

1. In the "Line Config" dialog box, select the limit line.
2. Select the "Edit" button.
3. Edit the line configuration as described in "How to configure a new limit line" on page 316.
4. Save the new configuration by selecting the "Save" button.

   If the limit line is active, the edited limit line is displayed in the diagram.

How to copy an existing limit line

1. In the "Line Config" dialog box, select the limit line.
2. Select the "Copy To" button.
3. Define a new name to create a new limit with the same configuration as the source line.
4. Edit the line configuration as described in "How to configure a new limit line" on page 316.
5. Save the new configuration by selecting the "Save" button.

   The new limit line is displayed in the overview and can be activated.

How to delete an existing limit line

1. In the "Line Config" dialog box, select the limit line.
2. Select the "Delete" button.
3. Confirm the message.

   The limit line and the results of the limit check are deleted.

How to configure a new limit line

1. In the "Line Config" dialog box, select the "New" button.

   The "Edit Limit Line" dialog box is displayed. The current line configuration is displayed in the preview area of the dialog box. The preview is updated after each change to the configuration.
2. Define a "Name" and, optionally, a "Comment" for the new limit line.
3. Define the x-axis configuration:
   - Time domain or frequency domain
   - Absolute or relative limits
   - Linear or logarithmic scaling

4. Define the y-axis configuration:
   - Level unit
   - Absolute or relative limits
   - Upper or lower limit line
5. Define the data points: minimum 2, maximum 200:
   a) Select "Insert Value".
   b) Define the x-value ("Position") and y-value ("Value") of the first data point.
   c) Select "Insert Value" again and define the second data point.
   d) Repeat this to insert all other data points.
   To insert a data point before an existing one, select the data point and then "Insert Value".
   To insert a new data point at the end of the list, move the focus to the line after the last entry and then select "Insert Value".
   To delete a data point, select the entry and then "Delete Value".
6. Check the current line configuration in the preview area of the dialog box. If necessary, correct individual data points or add or delete some.
   If necessary, shift the entire line vertically or horizontally by selecting the "Shift x" or "Shift y" button and defining the shift width.
7. Optionally, define a "Margin" at a fixed distance to the limit line.
   The margin must be within the valid value range and is not displayed in the diagram or preview area.
8. Optionally, if the y-axis uses relative scaling, define an absolute "Threshold" as an additional criteria for a violation.
9. Save the new configuration by selecting the "Save" button.

   The new limit line is displayed in the overview and can be activated.

How to move the limit line vertically or horizontally

A configured limit line can easily be moved vertically or horizontally. Thus, a new limit line can be easily generated based upon an existing limit line which has been shifted horizontally.

1. In the "Line Config" dialog box, select the limit line.
2. To shift the complete limit line parallel in the horizontal direction, select the "X-Offset" button and enter an offset value.
   To shift the complete limit line parallel in the vertical direction, select the "Y-Offset" button and enter an offset value.
3. To shift the individual data points of a limit line by a fixed value (all at once):
   a) Select the "Edit" button.
   b) In the "Edit Limit Line" dialog box, select the "Shift x" or "Shift y" button and define the shift width.
   c) Save the shifted data points by selecting the "Save" button.

   If activated, the limit line is shifted in the diagram.

7 Data Management

The R&S FSW allows you to store and load instrument settings, as well as import and export measurement data for analysis at a later time. Finally, you can store or print the measurement results displayed on the screen.

General storage and import/export functions are available via the toolbar. Some special storage functions are (also) available via softkeys or dialog boxes in the corresponding menus, e.g. trace data or marker peak lists.

7.1 Restoring the Default Instrument Configuration (Preset)

When delivered, the R&S FSW has a default configuration. You can restore this defined initial state at any time as a known starting point for measurements. This is often recommendable as a first step in troubleshooting when unusual measurement results arise.

To restore the default instrument configuration for all channels at once

► Press the PRESET key.

Alternatively to the factory default settings, you can define user-specific recall settings to be restored after a preset or reboot, see "To recall settings automatically after preset or reboot" on page 329.

After you use the PRESET function, the history of previous actions is deleted, i.e. any actions performed previously cannot be undone or redone using the UNDO/REDO keys.

SCPI command:

`*RST` or `SYSTem:PRESet`

To restore the default configuration for a single channel

The default measurement settings can also be reset for an individual channel only, rather than resetting the entire instrument.

► In the "Overview", select the "Preset Channel" button.

The factory default settings are restored to the current channel. Note that a user-defined recall settings file is **NOT** restored.

SCPI command:

SYSTem:PRESet:CHANnel[:EXECute] on page 690

7.1.1 Factory Default Configuration

The factory default configuration is selected such that the RF input is always protected against overload, provided that the applied signal levels are in the allowed range for the instrument.

Table 7-1: Factory default configuration

Parameter	Setting
mode	Spectrum
sweep mode	auto
center frequency	$f_{max}/2$
center frequency step size	0.1 * span
span	R&S FSW8: 8 GHz R&S FSW13: 13 GHz R&S FSW26: 26.5 GHz
RF attenuation	10 dB
reference level	0 dBm
level range	100 dB log
level unit	dBm
sweep time	auto
resolution bandwidth	auto (3 MHz)
video bandwidth	auto (3 MHz)
FFT filters	off
span/RBW	100
RBW/VBW	1
sweep	cont
trigger	free run
trace mode	1: clr write; 2/3/4/5/6: blank
detector	auto peak
frequency offset	0 Hz
reference level offset	0 dB
reference level position	100 %
grid	abs
cal correction	on

Parameter	Setting
noise source	off
input	RF

7.2 Storing and Recalling Instrument Settings and Measurement Data

Possibly you would like to restore or repeat a measurement you performed under specific conditions on the instrument. Or you want to evaluate imported data in another application on the R&S FSW and would like to restore the measurement settings applied during measurement. In these cases, you can store and recall instrument and measurement settings, and possibly other related measurement data.

Two different methods are available for managing instrument settings:

- Quick Save/Quick Recall - a defined set of instrument or channel settings are stored or recalled quickly in just one step
- Configurable Save/Recall - a user-defined set of instrument or channel settings are stored to a definable storage location

7.2.1 Quick Save/Quick Recall

The Quick Save and Quick Recall functions allow you to store instrument or channel settings very easily and quickly in one step. Up to 10 different sets of settings can be stored to or recalled from "save sets". Each save set is identified by its storage date and type (instrument or specific channel) in the display. The save sets are stored in the `C:\r_s\instr\user\QuickSave` directory, in files named `QuickSave1.dfl` to `QuickSave10.dfl`. The storage file names and locations cannot be changed.

During recall, save sets of type "Instrument" replace the settings of the entire instrument. All other save sets start a new measurement channel with the stored settings.

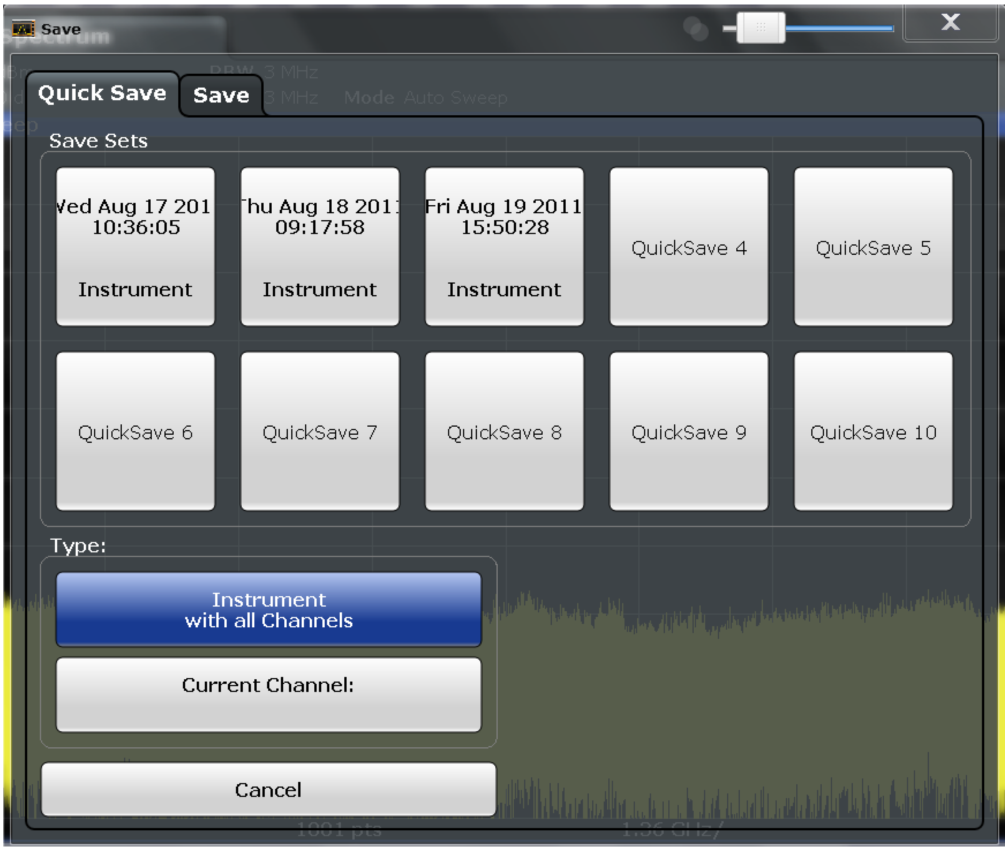
If a measurement channel with the same name as the channel to be restored is already active, the channel name for the new channel is extended by a consecutive number:

7.2.1.1 Quick Save / Quick Recall Dialog Boxes

The "QuickSave" dialog box is displayed when you select the "Save" icon in the toolbar.

The "Quick Recall" dialog box is displayed when you select the "Open" icon in the toolbar, or select the "Quick Recall" tab in the "Recall" dialog box.

Both dialog boxes are very similar and closely related.



QuickSave 1 / ... / QuickSave 10

Selects one of the save sets to store the current settings in or to be recalled. At the time of storage, the "QuickSave 1 / ... / QuickSave 10" placeholder is replaced by a label indicating the storage date and time and the storage type.

During recall, save sets of type "Instrument" replace the settings of the entire instrument. All other save sets start a new measurement channel with the stored settings.

Storage Type (Save only)

Defines which type of settings is to be stored in the save set.

"Instrument with all channels"	The instrument settings for all currently active channels are stored.
"Current Channel"	Only the instrument settings for the currently selected measurement channel are stored.

Recall

Restores the instrument settings as saved in the selected settings file. If the settings file contains settings for a specific channel only a new channel with the stored settings is activated, otherwise the entire instrument settings are loaded.

Note: After you use the "Recall" function, the history of previous actions is deleted, i.e. any actions performed previously cannot be undone or redone using the UNDO/REDO keys.

Note: If a measurement channel with the same name as the channel to be restored (in a new channel) is already active, the channel name for the new channel is extended by a consecutive number:

In remote commands, you must append this number to the channel name, as well.

SCPI command:

MMEMory:LOAD:STATe on page 688

Cancel

Closes the dialog box without saving the settings.

7.2.2 Configurable Storage and Recall

The more sophisticated storage and recall functions allow you to define which settings are stored, and where the settings file is stored to. Any settings file can be selected for recall.

7.2.2.1 Stored Data Types

The following types of data can be stored to and loaded from files via the "Save" dialog box on the R&S FSW:

Table 7-2: Items that can be stored to files

Item	Description
Current Settings	Current instrument and measurement settings
All Transducers	Transducer factors for all active transducers.
All Traces	All active traces; R&S FSW-K30 only: also calibration data

Item	Description
All Limit Lines	All limit lines (Note: information on which limit lines are active is stored with the "Current Settings")
Noise - ENR	Data in "ENR Settings" dialog box (R&S FSW-K30 only)
Noise - Loss Settings	Data in "Loss Settings" dialog box (R&S FSW-K30 only)
Noise - Calibration data	Results from calibration measurement (R&S FSW-K30 only)
K40 Results	All current phase noise trace results (R&S FSW-K40 only)

7.2.2.2 Storage Location and File Name

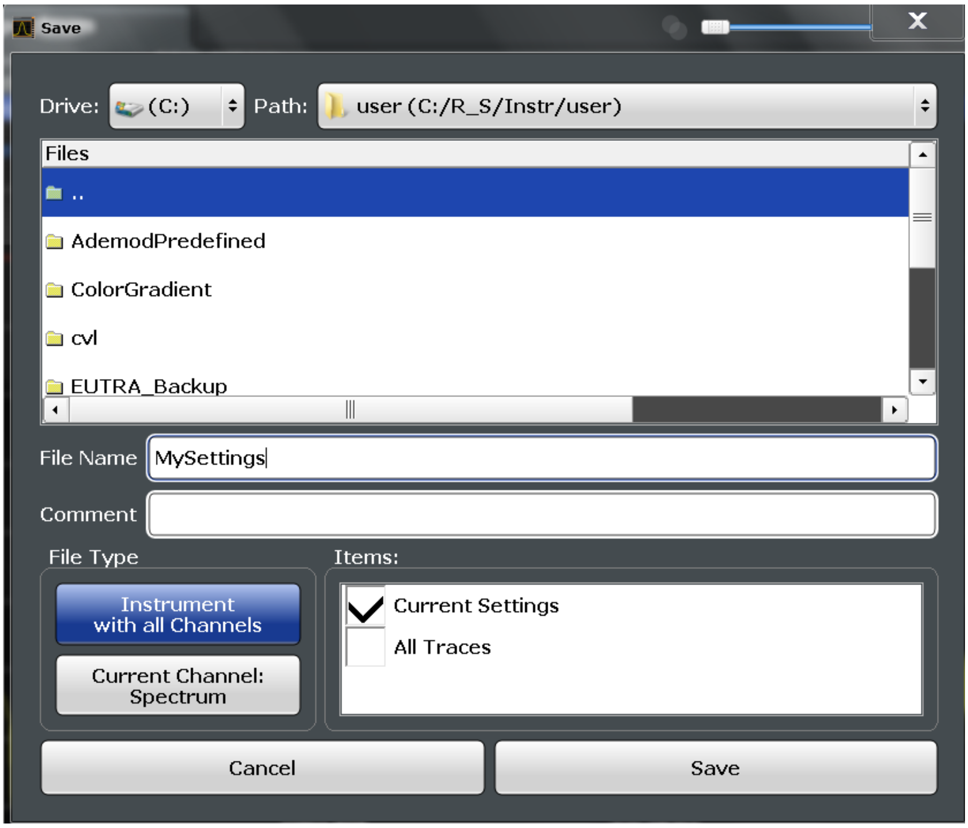
The data is stored on the internal flash disk or, if selected, on a memory stick or network drive. The operating system, firmware and stored instrument settings are located on drive C. All other folders and drives can be used to store measurement data.

The storage location and file name are selected in a file selection dialog box which is displayed when you perform a storage function.

By default, the name of a settings file consists of a base name followed by an underscore and three numbers, e.g. `limit_lines_005`. In the example, the base name is `limit_lines`. The base name can contain characters, numbers and underscores. The file extension `dfl` is added automatically. The default folder for settings files is `C:\r_s\instr\user`.

7.2.2.3 Save and Recall Dialog Boxes

The following dialog boxes are available via softkeys in the "Save/Recall" menu which is displayed when you select the "Save" or "Open" icon in the toolbar. Both dialog boxes are very similar and closely related.



Selecting the Storage Location - Drive/ Path/ Files

Select the storage location of the settings file on the instrument or an external drive.

The "Drive" indicates the internal (C:) or any connected external drives (e.g. a USB storage device).

The "Path" contains the drive and the complete file path to the currently selected folder.

The "Files" list contains all subfolders and files of the currently selected path.

The default storage location for the SEM settings files is: `C:\R_S\instr\sem_std`.

SCPI command:

`MMEMory:CATalog?` on page 678

File Name

Contain the name of the data file without the path or extension.

By default, the name of a settings file consists of a base name followed by an underscore. Multiple files with the same base name are extended by three numbers, e.g. `limit_lines_005`.

For details on the file name and location see chapter 7.2.2.2, "Storage Location and File Name", on page 323.

Comment

An optional description for the data file. A maximum of 60 characters can be displayed.

SCPI command:

`MMEMory:COMMent` on page 680

File Type

Determines whether the global instrument settings with all channels will be stored or recalled, or the current channel settings only.

Items

Defines which data and settings are stored or will be recalled. Depending on the "File Type", only channel settings or global settings are available. Which items are available also depends on the installed options (see also chapter 7.2.2.1, "Stored Data Types", on page 322).

SCPI command:

`MMEMory:SELect[:ITEM]:ALL` on page 684
`MMEMory:SELect[:ITEM]:DEFault` on page 684
`MMEMory:SELect[:ITEM]:HWSettings` on page 685
`MMEMory:SELect[:ITEM]:LINes:ALL` on page 685
`MMEMory:SELect[:ITEM]:NONE` on page 685
`MMEMory:SELect[:ITEM]:TRACe[:ACTive]` on page 686
`MMEMory:SELect[:ITEM]:TRANsducer:ALL` on page 686

Save File

Saves the settings file with the defined file name.

SCPI command:

`MMEMory:STORe:STATe` on page 688
`MMEMory:STORe:STATe:NEXT` on page 689

Recall in New Channel / Recall in Current Channel

Restores the instrument settings as saved in the selected settings file. If the settings file contains settings for a specific channel only, select "Recall in New Channel" to activate a new channel with the stored settings, or "Recall in Current Channel" to replace the current channel settings.

Note: After you use the "Recall" function, the history of previous actions is deleted, i.e. any actions performed previously cannot be undone or redone using the UNDO/REDO keys.

Note: If a measurement channel with the same name as the channel to be restored (in a new channel) is already active, the channel name for the new channel is extended by a consecutive number:

In remote commands, you must append this number to the channel name, as well.

SCPI command:

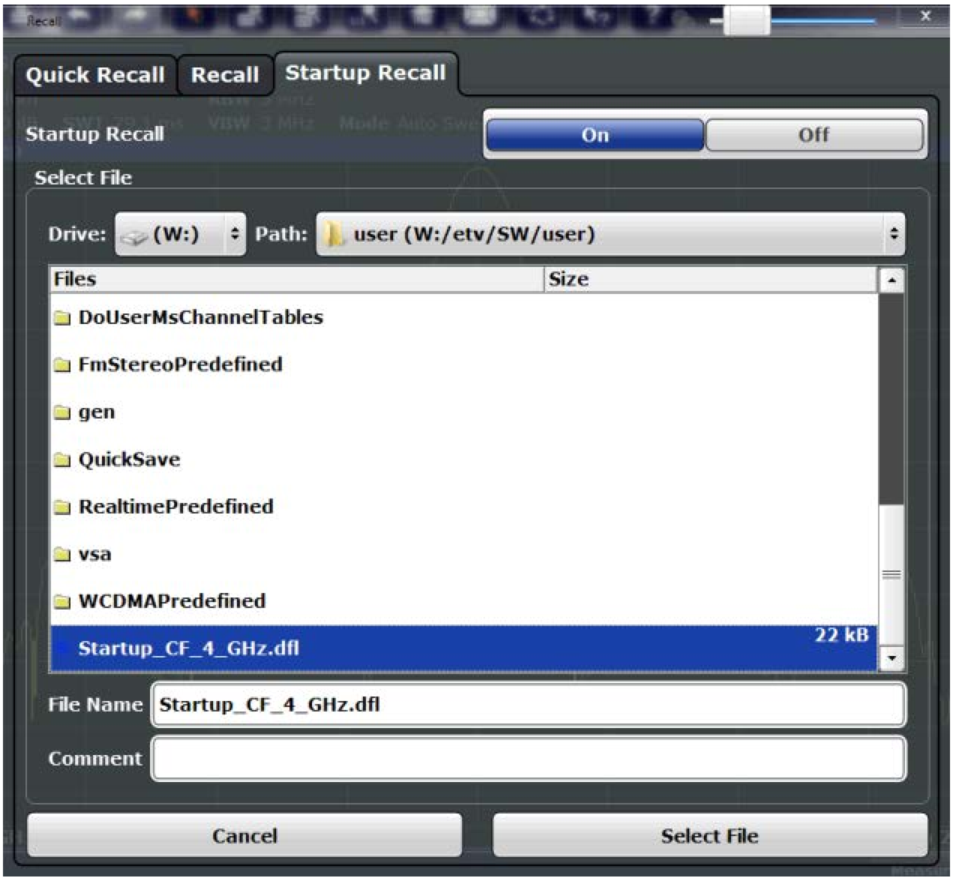
`MMEMory:LOAD:STATe` on page 688

Cancel

Closes the dialog box without saving the settings.

7.2.2.4 Startup Recall Settings

The "Startup Recall" softkey opens the "Startup Recall" tab of the "Recall" dialog box.



Startup Recall
Activates or deactivates the startup recall function. If activated, the settings stored in the selected file are loaded each time the instrument is started or preset. If deactivated, the default settings are loaded.

SCPI command:
`MMEMory:LOAD:AUTO` on page 687

Selecting the Storage Location - Drive/ Path/ Files
Select the storage location of the settings file on the instrument or an external drive.

The "Drive" indicates the internal (C:) or any connected external drives (e.g. a USB storage device).

The "Path" contains the drive and the complete file path to the currently selected folder.

The "Files" list contains all subfolders and files of the currently selected path.

The default storage location for the SEM settings files is: `C:\R_S\instr\sem_std`.

SCPI command:
`MMEMory:CATalog?` on page 678

File Name
Contain the name of the data file without the path or extension.

By default, the name of a settings file consists of a base name followed by an underscore. Multiple files with the same base name are extended by three numbers, e.g. `limit_lines_005`.

For details on the file name and location see chapter 7.2.2.2, "Storage Location and File Name", on page 323.

Comment
An optional description for the data file. A maximum of 60 characters can be displayed.

SCPI command:
`MMEMory:COMMent` on page 680

Cancel
Closes the dialog box without saving the settings.

7.2.3 How to Save and Load Instrument Settings

Instrument settings can be saved to a file and loaded again later, so that you can repeat the measurement with the same settings. Optionally, user-defined measurement settings can automatically be restored each time you start or preset the instrument.

To save and recall instrument settings using the Quick Save function

1. Select the  "Save" icon from the toolbar.
2. Select whether the instrument settings for **all** channels are to be stored, or only those for the **current** channel.
3. Select one of the save sets in which the settings are to be stored ("QuickSaveX").

The selected settings are stored to the file `C:\r_s\instr\user\QuickSave\QuickSaveX.dfl`.

4. To restore the settings, select the "Open" icon from the toolbar.
5. Select the save set in which the settings were stored ("QuickSaveX").

   The selected settings are restored to the instrument or channel.

To save configurable instrument settings

1. Select the "Save" icon from the toolbar.
2. In the "Save" dialog box, switch to the "Save" tab.
3. In the file selection dialog box, select a file name and storage location for the settings file.
4. Optionally, define a comment to describe the stored settings.
5. Select whether the instrument settings for **all** channels are to be stored, or only those for the **current** channel.
6. Select the items to be saved with the settings. Either the settings for the currently selected channel only or for all channels can be stored, and various other items such as lines or traces etc. can be stored as well (see chapter 7.2.2.1, "Stored Data Types", on page 322).
7. Select "Save".

   A file with the defined name and path and the extension `.dfl` is created.

To recall configurable instrument settings

1. Select the "Open" icon from the toolbar.
2. In the "Recall" dialog box, switch to the "Recall" tab.
3. In the file selection dialog box, select the file name and storage location of the settings file.

   Note: The "File Type" indicates whether the file contains instrument settings for **all** channels, or only those for the current channel.
4. If several items were saved, select which items are to be restored.
5. If channel settings were saved, select whether the settings will replace the settings in the current channel, or whether a new channel with the saved settings will be opened.
6. Select "Recall".

   The settings and selected items from the saved measurement are restored and you can repeat the measurement with the same settings.

To recall settings automatically after preset or reboot

You can define the settings that are restored when you preset or reboot the instrument.

1. Configure the settings as required and save them as described in "To save configurable instrument settings" on page 328.
2. In the "Save/Recall" menu, select the "Startup Recall" softkey.
3. If the file selection dialog box is not displayed automatically, select the "Select Dataset" softkey.
4. Select the recall settings that are to be restored.
5. Tap "Select".
6. Toggle the "Startup Recall" softkey to "On".

   Now when you press the PRESET key or reboot the instrument, the defined settings will be restored.
7. To restore the factory preset settings, toggle the "Startup Recall" softkey to "Off".

7.3 Importing and Exporting Measurement Results for Evaluation

The R&S FSW provides various evaluation methods for the results of the performed measurements. However, you may want to evaluate the data with further, external applications. In this case, you can export the measurement data to a standard format file (ASCII or XML). Some of the data stored in these formats can also be re-imported to the R&S FSW for further evaluation at a later time, for example in other applications.

The following data types can be exported:

- Trace data
- Marker peak lists
- I/Q data

The following data types can be imported:

- I/Q data

I/Q data can only be imported and exported in applications that process I/Q data, such as the I/Q Analyzer or optional applications.

See the corresponding user manuals for those applications for details.

7.3.1 Import/Export Functions

The following import and export functions are available via softkeys in the "Save/Recall" menu which is displayed when you select the "Save" or "Open" icon in the toolbar.

Some functions for particular data types are (also) available via softkeys or dialog boxes in the corresponding menus, e.g. trace data or marker peak lists.

For a description of the other functions in the "Save/Recall" menu see the R&S FSW User Manual.

Export
Opens a submenu to configure data export.

Export Trace to ASCII File ← Export
Opens a file selection dialog box and saves the selected trace in ASCII format (`.txt`) to the specified file and directory.

If the spectrogram display is selected when you perform this function, the entire histogram buffer with all frames is exported to a file. The data corresponding to a particular frame begins with information about the frame number and the time that frame was recorded. For large history buffers the export operation may take some time.

For details on the file format see chapter 7.3.4.1, "Reference: ASCII File Export Format", on page 332.

SCPI command:
`MMEMory:STORe<n>:TRACe` on page 697
`MMEMory:STORe:SGRam` on page 697

Decimal Separator ← Export
Defines the decimal separator for floating-point numerals for the data export files. Evaluation programs require different separators in different languages.

SCPI command:
`FORMat:DEXPort:DSEParator` on page 678

IQ Export ← Export
Opens a file selection dialog box to select an export file to which the IQ data will be stored. This function is only available in single sweep mode, and only in applications that process I/Q data, such as the I/Q Analyzer or optional applications.

For details see the description in the R&S FSW I/Q Analyzer User Manual ("Importing and Exporting I/Q Data").

Import
Provides functions to import data.

Currently, only I/Q data can be imported, and only by applications that process I/Q data.

See the R&S FSW I/Q Analyzer User Manual for more information.

7.3.2 How to Export Trace Data

The measured trace data can be exported to an ASCII file. For each sweep point the measured trace position and value are output. The file is stored with a `.DAT` extension.

For details on the storage format see chapter 7.3.4.1, "Reference: ASCII File Export Format", on page 332.

For the results of a Spectrum Emission Mask (SEM) or Spurious Emissions measurement, special file export functions are available, see chapter 4.5.6.2, "How to Save SEM Result Files", on page 97(SEM) and "Saving the Evaluation List" on page 114 (Spurious).

Trace data can be exported either from the "Trace" menu, or from the "Save/Recall" menu.

To export from the "Save/Recall" menu

1. Select an active trace whose data you want to export.
2. Select the "Save" icon in the toolbar.
3. Select the "Export" softkey.
4. If necessary, change the decimal separator to be used for the ASCII export file.
5. Select the "Trace ASCII Export" softkey.
6. In the file selection dialog box, select the storage location and file name for the export file.
7. Select "Save" to close the dialog box and export the trace data to the file.

To export from the "Trace" menu

1. Press the TRACE key.
2. Select the "Trace Config" softkey.
3. Select the "Trace Export" tab.
4. Select an active trace whose data you want to export.
5. If necessary, change the decimal separator to be used for the ASCII export file.
6. Select the "Trace ASCII Export" button.
7. In the file selection dialog box, select the storage location and file name for the export file.

8. Select "Save" to close the dialog box and export the trace data to the file.

7.3.3 How to Export a Peak List

You can save the results of a marker peak list to an ASCII file.

1. Press the MKR FUNCT key.
2. Select the "Marker Peak List" softkey.
3. Configure the peak search and list settings as described in "Marker Peak List Configuration" on page 298.
4. Set the marker peak list "State" to "On".
5. Press the RUN SINGLE key to perform a single sweep measurement and create a marker peak list.
6. Select the "Marker Peak List" softkey to display the "Marker Peak List" dialog box again.
7. If necessary, change the decimal separator to be used for the ASCII export file.
8. Select the "Export Peak List" button.
9. In the file selection dialog box, select the storage location and file name for the export file.
10. Select "Save" to close the dialog box and export the peak list data to the file.

7.3.4 Reference: File Format Descriptions

This reference describes in detail the format of the export files for result data.

For a description of the file formats for spectrum emission mask (SEM) measurement settings and results, see chapter 4.5.7, "Reference: SEM File Descriptions", on page 98.

The file format for Spurious Emissions measurement results is described in chapter 4.6.6, "Reference: ASCII Export File Format (Spurious)", on page 115.

7.3.4.1 Reference: ASCII File Export Format

Trace data can be exported to a file in ASCII format for further evaluation in other applications

(For details see chapter 7.3.2, "How to Export Trace Data", on page 331).

The file consists of the header containing important scaling parameters and a data section containing the trace data.

Generally, the format of this ASCII file can be processed by spreadsheet calculation programs, e.g. MS-Excel. Different language versions of evaluation programs may require a different handling of the decimal point. Thus you can define the decimal separator to be used (decimal point or comma, see "Decimal Separator" on page 255).

The data of the file header consist of three columns, each separated by a semicolon: parameter name; numeric value; basic unit. The data section starts with the keyword "Trace <n>" (<n> = number of stored trace), followed by the measured data in one or several columns (depending on measurement) which are also separated by a semicolon.

If the spectrogram display is selected when you select the "ASCII Trace Export" softkey, the entire histogram buffer with all frames is exported to a file. The data corresponding to a particular frame begins with information about the frame number and the time that frame was recorded.

Table 7-3: ASCII file format for trace export

File contents	Description
Header data	
Type;R&S FSW;	Instrument model
Version;5.00;	Firmware version
Date;01.Oct 2006;	Date of data set storage
Mode;ANALYZER;	Operating mode
Center Freq;55000;Hz	Center frequency
Freq Offset;0;Hz	Frequency offset
Span;90000;Hz	Frequency range (0 Hz in zero span and statistics measurements)
x-Axis;LIN;	Scaling of x-axis linear (LIN) or logarithmic (LOG)
Start;10000;Hz Stop;100000;Hz	Start/stop of the display range. Unit: Hz for span > 0, s for span = 0, dBm/dB for statistics measurements
Ref Level;-30;dBm	Reference level
Level Offset;0;dB	Level offset
Ref Position;75; %	Position of reference level referred to diagram limits (0 % = lower edge)
y-Axis;LOG;	Scaling of y-axis linear (LIN) or logarithmic (LOG)
Level Range;100;dB	Display range in y direction. Unit: dB with x-axis LOG, % with x-axis LIN
Rf Att;20;dB	Input attenuation
El Att;2.0;dB	
RBW;100000;Hz	Resolution bandwidth
VBW;30000;Hz	Video bandwidth

File contents	Description
SWT;0.005;s	Sweep time
Trace Mode;AVERAGE;	Display mode of trace: CLR/WRITE,AVERAGE,MAX-HOLD,MINHOLD
Detector;AUTOPEAK;	Detector set: AUTOPEAK,MAXPEAK,MINPEAK,AVER-AGE,RMS,SAMPLE,QUASIPEAK
Sweep Count;20;	Number of sweeps set
Preamplifier;OFF	Preamplifier status
Data section	
Trace 1:;;	Selected trace
x-Unit;Hz;	Unit of x values: Hz with span > 0; s with span = 0; dBm/dB with statistics measurements
y-Unit;dBm;	Unit of y values: dB*/V/A/W depending on the selected unit with y-axis LOG or % with y-axis LIN
Values; 1001;	Number of measurement points
10000;-10.3;-15.7 10130;-11.5;-16.9 10360;-12.0;-17.4 ...;...;	Measured values: <x value>, <y1>, <y2>; <y2> being available only with detector AUTOPEAK and containing in this case the smallest of the two measured values for a measurement point.

Table 7-4: ASCII file format for spectrogram trace export

File contents	Description
Header	
Type;R&S FSW;	Instrument model
Version;5.00;	Firmware version
Date;01.Oct 2006;	Date of data set storage
Mode;ANALYZER;**SPECTROGRAM**	Operating mode
Center Freq;55000;Hz	Center frequency
Freq Offset;0;Hz	Frequency offset
Span;90000;Hz	Frequency range (0 Hz in zero span and statistics measure-ments)
x-Axis;LIN;	Scaling of x-axis linear (LIN) or logarithmic (LOG)
Start;10000;Hz Stop;100000;Hz	Start/stop of the display range. Unit: Hz for span > 0, s for span = 0, dBm/dB for statistics meas-urements
Ref Level;-30;dBm	Reference level
Level Offset;0;dB	Level offset
Ref Position;75; %	Position of reference level referred to diagram limits (0 % = lower edge)

File contents	Description
y-Axis;LOG;	Scaling of y-axis linear (LIN) or logarithmic (LOG)
Level Range;100;dB	Display range in y direction. Unit: dB with x-axis LOG, % with x-axis LIN
Rf Att;20;dB	Input attenuation
RBW;100000;Hz	Resolution bandwidth
VBW;30000;Hz	Video bandwidth
SWT;0.005;s	Sweep time
Trace Mode;AVERAGE;	Display mode of trace: CLR/WRITE,AVERAGE,MAX-HOLD,MINHOLD
Detector;AUTOPEAK;	Detector set: AUTOPEAK,MAXPEAK,MINPEAK,AVER-AGE,RMS,SAMPLE,QUASIPEAK
Sweep Count;20;	Number of sweeps set
Data section	
Trace 1:;;	Selected trace
x-Unit;Hz;	Unit of x values: Hz with span > 0; s with span = 0; dBm/dB with statistics measurements
y-Unit;dBm;	Unit of y values: dB*/V/A/W depending on the selected unit with y-axis LOG or % with y-axis LIN
Values; 1001;	Number of measurement points
Frames;2;	**Number of exported frames**
Frame;0;	**Most recent frame number**
Timestamp;17.Mar 11;11:27:05.990	**Timestamp of this frame**
10000;-10.3;-15.7 10130;-11.5;-16.9 10360;-12.0;-17.4 ...;...;	Measured values, identical to spectrum data: <x value>, <y1>, <y2>; <y2> being available only with detector AUTOPEAK and containing in this case the smallest of the two measured values for a measurement point.
Frame;-1;	**Next frame**
Timestamp;17.Mar 11;11:27:05.342	**Timestamp of this frame**
...	

7.4 Creating Screenshots of Current Measurement Results and Settings

In order to document the graphical results and the most important settings for the currently performed measurement, you can create a hardcopy or screenshot of the current display. Screenshots can either be printed or stored to a file.

7.4.1 Print and Screenshot Settings

The settings for saving and printing screenshots are configured via the "Print" menu which is displayed when you select the "Print" icon in the toolbar.

For step-by-step instructions see chapter 7.4.2, "How to Store or Print Screenshots of the Display", on page 339.

Remote commands for these settings are described in chapter 10.7.4, "Storing or Printing Screenshots", on page 690.

To print a screensot of the current display with the current settings immediately, without switching to the "Print" menu, use the "Print immediately" icon at the right-hand side of the toolbar.

Printing or Storing a Screenshot (Print Screen)
Starts to print out or store all measurement results displayed on the screen: diagrams, traces, markers, marker lists, limit lines, etc., including the channel and status bars. Optionally, comments and the date and time are included at the bottom margin of the printout. All displayed items belonging to the software user interface (e.g. softkeys or dialog boxes) are not printed out.

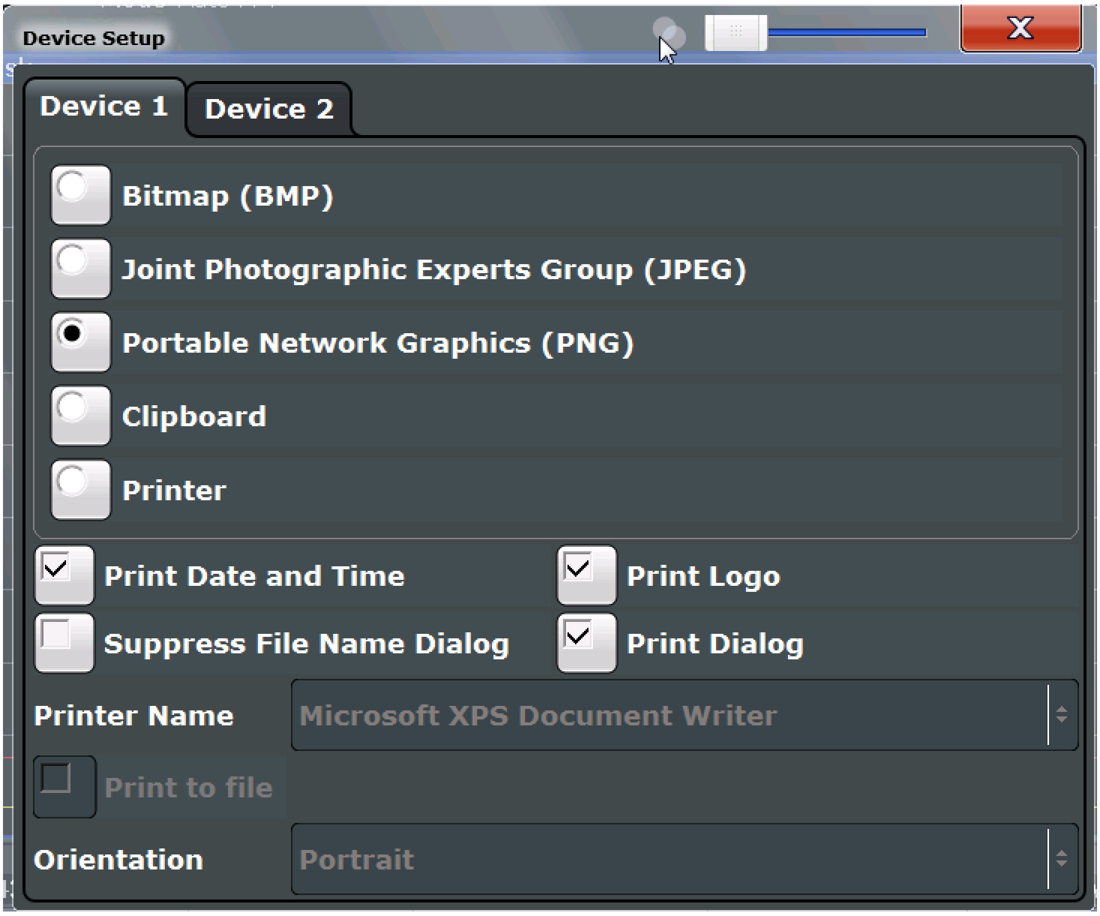
Whether the output is sent to the printer or stored in a file or the clipboard depends on the selected device and the device settings.

If the output is stored to a file, a file selection dialog box is opened to select the file name and location. The default path is `C:\r_s\instr\user`.

The "Print" menu is displayed to configure printing.

SCPI command:
`HCOPy:ITEM:ALL` on page 694
`HCOPy[:IMMediate<device>]` on page 693
`HCOPy[:IMMediate<device>]:NEXT` on page 694

Device Setup
Defines the behavior of the "Print Screen" function, depending on which device is selected. Two different devices can be configured, e.g. one for printing and one for storage to a file.



Output Medium ← Device Setup
Defines the medium to which the screenshot is printed or stored.

"File formats" Stores the screenshot to a file in the selected format. The file name is queried at the time of storage.

"Clipboard" Stores the screenshot to the clipboard.

"Printer" Prints the screenshot on the printer selected from the "Name" list.

SCPI command:
HCOPy:DEVice:LANGuage<device> on page 693

Print Date and Time ← Device Setup
Activates/deactivates the printout of the current date and time at the bottom of the screenshot.

Print Logo ← Device Setup
Activates/deactivates the printout of the Rohde & Schwarz company logo in the upper left corner.

Suppress File Name Dialog ← Device Setup
When the screenshot is stored to a file, the file selection dialog box is not displayed. Instead, the current storage location and file name are used (default: C:\r_S\instr\user). Each new the file name is extended by a consecutive number, e.g. File002, File003 etc.

Print Dialog ← Device Setup
Includes any currently displayed dialog in the screenshot.

Printer Name ← Device Setup
Defines the printer to print to.

SCPI command:
SYSTem:COMMunicate:PRINter:ENUMerate[:NEXT]? on page 695
SYSTem:COMMunicate:PRINter:ENUMerate:FIRSt? on page 695
SYSTem:COMMunicate:PRINter:SELect<device> on page 695

Print to File ← Device Setup
If a printer is selected as the output medium, use this option to store the data in a .prn file using the selected printer driver.

Orientation ← Device Setup
Selects the page orientation of the printout: portrait or landscape (printer only)

SCPI command:
HCOPy:PAGE:ORIentation<device> on page 694

Device
Two different printout devices can be configured, e.g. one for printing and one for storage to a file. When the "Print Screen" function is executed, the selected device and its settings determine the behavior.

SCPI command:
HCOPy:DESTination<device> on page 692

Colors
Opens the "Print Color" dialog box to configure the colors for printing screenshots. For details see chapter 8.4.3, "Display Theme and Colors", on page 369.

Comment
Defines an optional comment to be printed with the screenshot of the display. Maximum 120 characters are allowed. 60 characters fit in one line. In the first line, at any point a manual line-feed can be forced by entering "@".

Date and time are inserted automatically. The comment is printed below the diagram area, but not displayed on the screen. If a comment should not be printed, it must be deleted.

SCPI command:
`HCOPy:ITEM:WINDow:TEXT` on page 694

Install Printer
Opens the standard Windows dialog box to install a new printer. All printers that are already installed are displayed.

Only user accounts with administrator rights can install a printer.

For further information refer to the Microsoft Windows documentation.

7.4.2 How to Store or Print Screenshots of the Display

► If the R&S FSW has already been set up according to your current requirements, simply press the "Print immediate" icon ( ) at the far right end of the toolbar.

A screenshot of the current measurement display is printed or stored to a file, as configured.

To set up screenshot outputs

This configuration assumes a printer has already been installed. To install a new printer, use the Install Printer softkey.

1. Select the "Printer" tool in the toolbar.
2. Select the "Device Setup" softkey.
3. Select the tab for Device 1 or Device 2 to configure a device.
4. To set up the print function to store a screenshot to a file, select the required file format as the output medium.
   To set up the print function to store a screenshot to the clipboard, select "Clipboard" as the output medium.
   To set up the print function to print a screenshot on a printer, select "Printer" as the output medium and an installed printer from the "Name" list.
5. For printout, select the page orientation.
6. Optionally, deactivate the date and time or the logo so they are not added to the screenshot.
7. Select "OK" to close the "Device Setup" dialog box.
8. Toggle the "Device" softkey to the device configuration you want to use.

9. Optionally, configure the colors to be used for printout, e.g. as displayed on the screen instead of inversed.
10. Optionally, add a comment to be included with the screenshot.
11. Select the "Print Screen" softkey or the "Printer" or "Screenshot" tool in the toolbar to execute the print function and check the results.
12. If you configured the print function to store the screenshot to a file, enter a file name in the file selection dialog box.

8 General Instrument Setup

Some basic instrument settings can be configured independantly of the selected operating mode or application. Usually, you will configure most of these settings initially when you set up the instrument according to your personal preferences or requirements and then only adapt individual settings to special circumstances when necessary. Some special functions are provided for service and basic system configuration.

8.1 Basics on Alignment

When you put the instrument into operation for the first time or when strong temperature changes occur, it may be necessary to align the data to a reference source (see also "Temperature check" on page 342).

During instrument start, the installed hardware is checked against the current firmware version to ensure the hardware is supported. If not, an error message is displayed ("WRONG_FW") and you are asked to update the firmware. Until the firmware version is updated, self-alignment fails.

The correction data and characteristics required for the alignment are determined by comparison of the results at different settings with the known characteristics of the high-precision calibration signal source at 64 MHz.

Alignment results

The alignment results are displayed and contain the following information:

- date and time of last correction data record
- overall results of correction data record
- list of found correction values according to function/module

The results are classified as follows:

PASSED	Calibration successful without any restrictions
CHECK	Deviation of correction value larger than expected, correction could however be performed
FAILED	Deviations of correction value too large, no correction was possible. The found correction data is not applicable.

The results are available until the next self-alignment process is started or the instrument is switched off.

Temperature check

During self-alignment, the instrument's (frontend) temperature is also measured (as soon as the instrument has warmed up completely). This temperature is used as a reference for a continuous temperature check during operation. If the current temperature deviates from the stored self-alignment temperature by a certain degree, a warning is displayed in the status bar indicating the resulting deviation in the measured power levels. A status bit in the `STATUs:QUEStionable:TEMPerature` register indicates a possible deviation. The current temperature of the RF Frontend can be queried using a remote command (see `SOURce:TEMPerature:FRONtend?` on page 704).

Touch screen alignment

When the device is delivered, the touch screen is initially calibrated. However, to ensure that the touch screen responds to the finger contact correctly, a touch screen alignment is required.

Alignment of the touch screen is useful:

- At first use
- After an image update or after exchanging a hard disk
- If you notice that touching a specific point on the screen does not achieve the correct response
- If the position of the instrument has been changed and you cannot look straight on the screen
- If another person operates the instrument

8.2 Basics on Transducer Factors

The transducer allows you to manipulate the trace at discrete trace points to correct the signal coming from an input device. Transducers are often used to correct the frequency response for antennas, for example. The transducer is configured by defining transducer factors for specific trace points. A set of transducer factors defines an interpolated transducer line and can be stored on the instrument.

In the Spectrum application, the correction factor from all active transducers is calculated for each displayed trace point once in advance and is added to the result of the level measurement during the sweep. If the sweep range changes, the correction values are calculated again. If several measured values are combined in one point, only one value is taken into consideration. If the active transducer line is not defined for the entire sweep range, the missing values are replaced by zeroes.

When a transducer is used, the trace is shifted by a calculated factor. However, an upward shift reduces the dynamic range for the displayed values. Thus, the reference level can be adapted automatically to restore the original dynamic range. The reference level is shifted by the maximum transducer factor. By default, if transducers are active the reference level function is adapted automatically to obtain the best dynamic performance.

If a transducer factor is active, "TDF" is displayed in the channel bar.

Y-Axis Unit

The individual transducer factors can be defined as absolute values or relative (dB) values. However, all factors for one transducer line use the same unit. As soon as a transducer is activated, the unit of the transducer is automatically used for all the level settings and outputs. The unit cannot be changed in the amplitude settings since the R&S FSW and the active transducer are regarded as one measuring instrument. Only for relative transducer factors (unit dB), the unit originally set on the instrument is maintained and can be changed.

When all transducers have been switched off, the R&S FSW returns to the unit that was used before a transducer was activated.

Configuration

The R&S FSW supports transducer lines with a maximum of 1001 data points. Eight of the transducer lines stored in the instrument can be activated simultaneously. The number of transducer lines stored in the instrument is only limited by the capacity of the storage device used.

A transducer line consists of the following data:

- A maximum of 1001 data points with a position and value
- A unit for the values
- A name to distinguish the transducer lines

Validity

The transducer factors must comply with the following rules to ensure correct operation:

- The frequencies for the data points must always be defined in ascending order. Otherwise the entry will not be accepted and the an error message is displayed.
- The frequencies of the data points may exceed the valid frequency range of the R&S FSW since only the set frequency range is taken into account for measurements. The minimum frequency of a data point is 0 Hz, the maximum frequency 200 GHz.
- The value range for the transducer factor is ±200 dB.
- Gain has to be entered as a negative value, and attenuation as a positive value.

8.3 General Instrument Settings

Instrument settings can be configured via the SETUP key.

Network and Remote Settings, Display Settings

Settings for network and remote operation are described in chapter 9, "Network and Remote Operation", on page 381.

Display settings are described in chapter 8.4, "Display Settings", on page 363.

8.3.1 Reference Frequency Settings

The reference frequency settings are defined in the "Reference" dialog box which is displayed when you press the SETUP key and then select "Reference".

Reference Frequency Input

The R&S FSW can use the internal reference source or an external reference source as the frequency standard for all internal oscillators. A 10 MHz crystal oscillator is used as the internal reference source. In the external reference setting, all internal oscillators of the R&S FSW are synchronized to the external reference frequency. External references are connected to one of the REF INPUT or the SYNC TRIGGER connectors on the rear panel. For details see the "Getting Started" manual.

Note: Optionally (R&S FSW-B4), the more precise OCXO signal can replace the internal reference source.

The default setting is the internal reference. When an external reference is used, "EXT REF" is displayed in the status bar.

The following reference inputs are available:

Table 8-1: Available Reference Frequency Input

Source	Frequency	Tuning Range	Loop Band-width	Description
Internal	10 MHz	-	1-100 Hz	Internal reference signal or OCXO (option R&S FSW-B4)
External Reference 10 MHz	10 MHz	+/- 6 ppm	1-100 Hz	External reference from REF INPUT 1..20 MHZ connector; Fixed external 10 MHZ reference fre-quency. Good phase noise performance
External Reference 1..20 MHz	1..20 MHz in 1 Hz steps	+/- 0.5 ppm	0.1 Hz (fixed)	Variable external reference frequency in 0.1 Hz steps from REF INPUT 1..20 MHZ connector; Good external phase noise suppression. Small tuning range.
		+/- 6 ppm	1-30 Hz	Variable external reference frequency in 0.1 Hz steps from REF INPUT 1..20 MHZ connector; Wide tuning range.
External Reference 100 MHz	100 MHz	+/- 6 ppm	1-300 Hz	External reference from REF INPUT 100 MHZ connector Good phase noise performance
Sync Trigger	100 MHz	+/- 6 ppm	1-300 Hz	External reference from SYNC TRIGGER INPUT connector

SCPI command:

[SENSe:]ROSCillator:SOURce on page 701
SOURce:EXTernal:ROSCillator:EXTernal:FREQuency on page 701

Behavior in case of missing external reference ← Reference Frequency Input

If an external reference is selected but none is available, there are different ways the instrument can react.

"Show Error Flag" The message "NO REF" is displayed to indicate that no synchronization is performed.

"Switch to internal reference" The instrument automatically switches back to the internal reference if no external reference is available. Note that you must re-activate the external reference if it becomes available again at a later time.

SCPI command:
[SENSe:]ROSCillator:SOURce on page 701
[SENSe:]ROSCillator:SOURce:EAUTo? on page 702

Tuning Range ← Reference Frequency Input
The tuning range is only available for the variable external reference frequency. It determines how far the frequency may deviate from the defined level in parts per million (10^{-6}).

"+/- 0.5 ppm" With this smaller deviation a very narrow fixed loop bandwidth of 0.1 Hz is realized. With this setting the instrument can synchronize to an external reference signal with a very precise frequency. Due to the very narrow loop bandwidth, unwanted noise or spurious components on the external reference input signal are strongly attenuated. Furthermore, the loop requires about 30 seconds to reach a locked state. During this locking process, "NO REF" is displayed in the status bar.

"+/- 6 ppm" The larger deviation allows the instrument to synchronize to less precise external reference input signals.

Frequency ← Reference Frequency Input
Defines the external reference frequency to be used (for variable connectors only).

Loop Bandwidth ← Reference Frequency Input
Defines the speed of internal synchronization with the reference frequency. The setting requires a compromise between performance and increasing phase noise.

For a variable external reference frequency with a narrow tuning range (+/- 0.5 ppm), the loop bandwidth is fixed to 0.1 Hz and cannot be changed.

Reference Frequency Output
A reference frequency can be provided by the R&S FSW to other devices that are connected to this instrument. If one of the following options is activated, the reference signal is output to the corresponding connector.

"Output 100 MHz"
Provides a 100 MHz reference signal to the REF OUTPUT 100 MHZ connector.

"Ouput 640 MHz"
Provides a 640 MHz reference signal to the REF OUTPUT 640 MHZ connector.

"Output Sync Trigger"
Provides a 100 MHz reference signal to the SYNC TRIGGER OUTPUT connector.

Resetting the Default Values

The values for the "Tuning Range", "Frequency" and "Loop Bandwidth" are stored for each source of "Reference Frequency Input". Thus, when you switch the input source, the previously defined settings are restored. You can restore the default values for all input sources using the "Preset Channel" function.

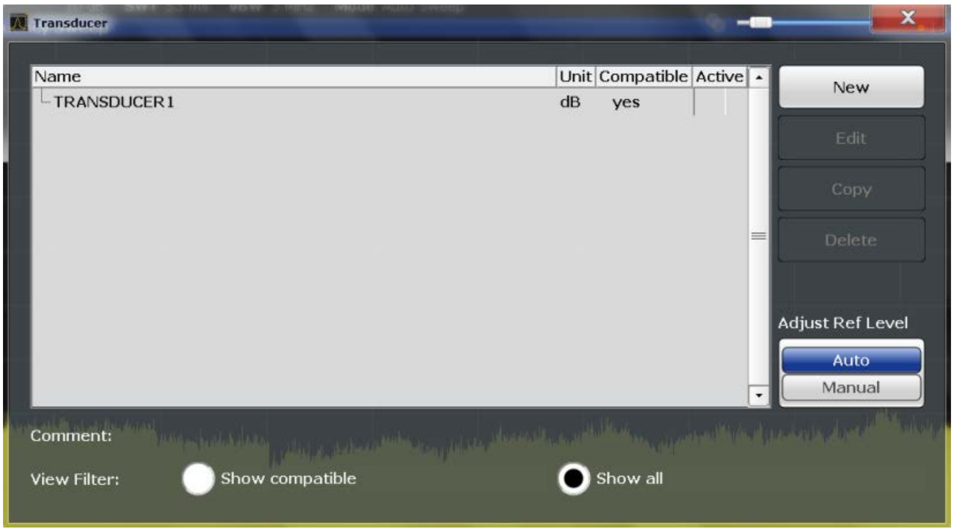
8.3.2 Transducer Settings

Up to 8 transducer lines can be activated simultaneously in the R&S FSW. Many more can be stored on the instrument.

The transducer settings are defined in the "Transducer" dialog box which is displayed when you press the SETUP key and then select "Transducer".

8.3.2.1 Transducer Management

The settings required to manage all transducer lines on the instrument are described here.



For the transducer line overview, the R&S FSW searches for all stored transducer lines with the file extension `.TDF` in the `trd` subfolder of the main installation folder. The overview allows you to determine which transducer lines are available and can be used for the current measurement.

For details on settings for individual lines see chapter 8.3.2.2, "Transducer Factors", on page 349.

For instructions on configuring and working with transducers see chapter 8.6.5, "How to Configure the Transducer", on page 376.

Name

The name of the stored transducer line.

Unit

The unit in which the y-values of the data points of the transducer line are defined.

The following units are available:

- dB
- dBm
- dBmV
- dBμV
- dBμV/m
- dBμA
- dBμA/m
- dBpW
- dBpT

Compatibility

Indicates whether the transducer factors are compatible with the current measurement settings.

For more information on which conditions a transducer line must fulfill to be compatible, see chapter 8.2, "Basics on Transducer Factors", on page 342.

Activating/Deactivating

Activates/deactivates the transducer line. Up to 8 transducer lines can be active at the same time.

SCPI command:

`[SENSe:]CORRection:TRANsducer:SELect` on page 706

`[SENSe:]CORRection:TRANsducer[:STATe]` on page 706

Comment

An optional description of the transducer line.

Included Lines in Overview (View Filter)

Defines which of the stored lines are included in the overview. The view can be restricted to compatible lines only or include all lines found. Whether a line is compatible or not is indicated in the Compatibility setting.

Adjusting the Reference Level

Activates or deactivates the automatic adjustment of the reference level to the selected transducer factor.

"Auto" Activates the automatic adjustment. The original dynamic range is restored by shifting the reference level by the maximum transducer factor.

"Man" Deactivates the automatic adjustment. Adjust the reference level via the "Amplitude" menu.

SCPI command:

`[SENSe:]CORRection:TRANsducer:ADJust:RLEVel[:STATe]` on page 705

Create New Line

Create a new transducer line.

SCPI command:

`[SENSe:]CORRection:TRANsducer:SELect` on page 706

Edit Line

Edit an existing transducer line configuration.

Copy Line

Copy the selected transducer line configuration to create a new line.

Delete Line

Delete the selected transducer line.

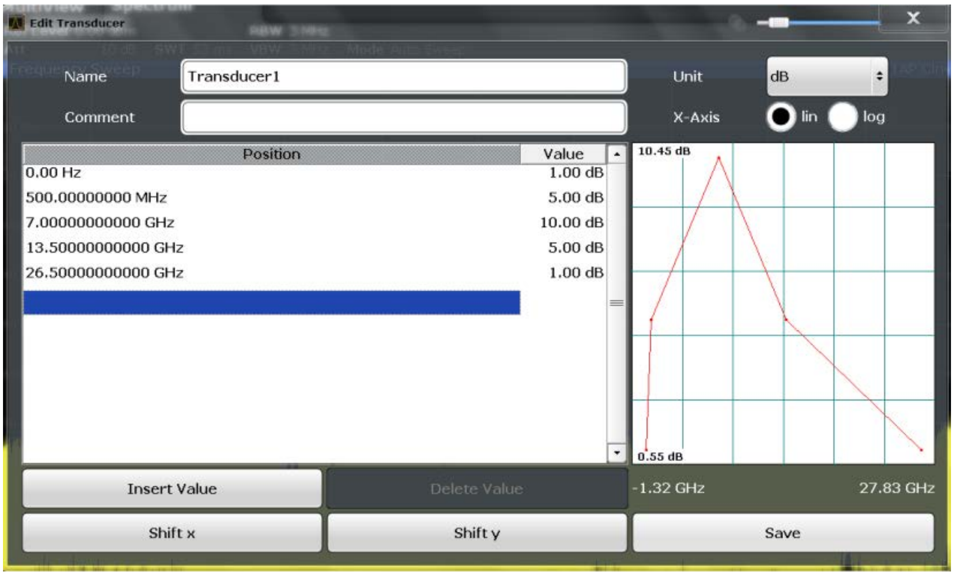
SCPI command:

`[SENSe:]CORRection:TRANsducer:DELete` on page 706

8.3.2.2 Transducer Factors

The settings and functions available for individual transducer lines are described here.

For instructions on creating and editing transducer lines see chapter 8.6.5, "How to Configure the Transducer", on page 376.



Name

Defines the transducer line name. All names must be compatible with the Windows7 conventions for file names. The transducer data is stored under this name (with a `.TDF` extension) in the `trd` subfolder of the main installation folder.

SCPI command:

`[SENSe:]CORRection:TRANsducer:SELect` on page 706

Comment

Defines an optional comment for the transducer line. The text may contain up to 40 characters.

SCPI command:

`[SENSe:]CORRection:TRANsducer:COMMent` on page 705

Unit

The unit in which the y-values of the data points of the transducer line are defined.

As soon as a transducer is activated, the unit of the transducer is automatically used for all the level settings and outputs. The unit cannot be changed in the amplitude settings unless dB is used.

SCPI command:

`[SENSe:]CORRection:TRANsducer:UNIT` on page 707

X-Axis Scaling
Describes the scaling of the horizontal axis on which the data points of the transducer line are defined. Scaling can be linear or logarithmic.

SCPI command:
`[SENSe:]CORRection:TRANsducer:SCALing` on page 706

Data points
Each transducer line is defined by a minimum of 2 and a maximum of 50 data points. Each data point is defined by its position (x-axis) and value (y-value).

The data points must comply with the following rules to ensure correct operation:
- The frequencies for the data points must always be defined in ascending order. Otherwise the entry will not be accepted and the an error message is displayed.
- The frequencies of the data points may exceed the valid frequency range of the R&S FSW since only the set frequency range is taken into account for measurements. The minimum frequency of a data point is 0 Hz, the maximum frequency 200 GHz.
- The value range for the transducer factor is ±200 dB.
- Gain has to be entered as a negative value, and attenuation as a positive value.

SCPI command:
`[SENSe:]CORRection:TRANsducer:DATA` on page 705

Insert Value
Inserts a data point in the transducer line above the selected one in the "Edit Transducer" dialog box.

Delete Value
Deletes the selected data point in the "Edit Transducer" dialog box.

Shift x
Shifts the x-value of each data point horizontally by the defined shift width.

Shift y
Shifts the y-value of each data point vertically by the defined shift width.

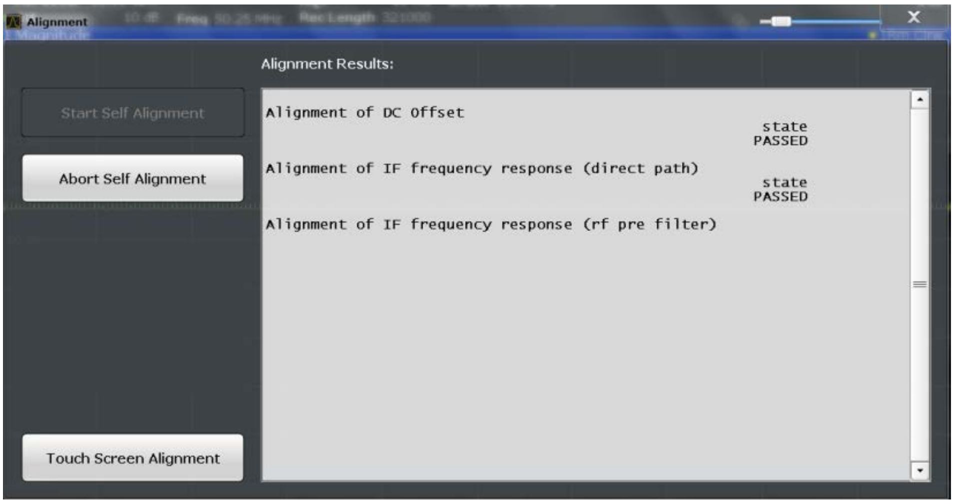
Save
Saves the currently edited transducer line under the name defined in the "Name" field.

SCPI command:
`MMEMory:SELect[:ITEM]:TRANsducer:ALL` on page 686
`MMEMory:STORe:STATe` on page 688

8.3.3 Alignment Settings

Both the instrument and the touch screen can be aligned when necessary (see chapter 8.1, "Basics on Alignment", on page 341).

The alignment settings are defined in the "Alignment" dialog box which is displayed when you press the SETUP key and then select "Alignment".



Starting a Self-alignment

Starts recording correction data for the instrument. If the correction data acquisition fails or if the correction values are deactivated, a corresponding message is displayed in the status field.

For details see chapter 8.1, "Basics on Alignment", on page 341.

Note:

A running Sequencer operation is aborted when you start a self-alignment.

SCPI command:

`*CAL?` on page 445

Aborting the Self-alignment

As long as the self-alignment data is being collected the procedure can be cancelled using the "Abort Self-alignment" button.

SCPI command:

`ABORt` on page 459

Starting the Touch Screen Alignment

Starts the touch screen alignment.

Tap the 4 markers on the screen as you are asked to do. The touch screen is aligned according to the executed pointing operations.

Alignment Results

Information on whether the alignment was performed succesfully and on the applied correction data is displayed. The results are available until the next self-alignment process is started or the instrument is switched off.

8.3.4 System Configuration Settings

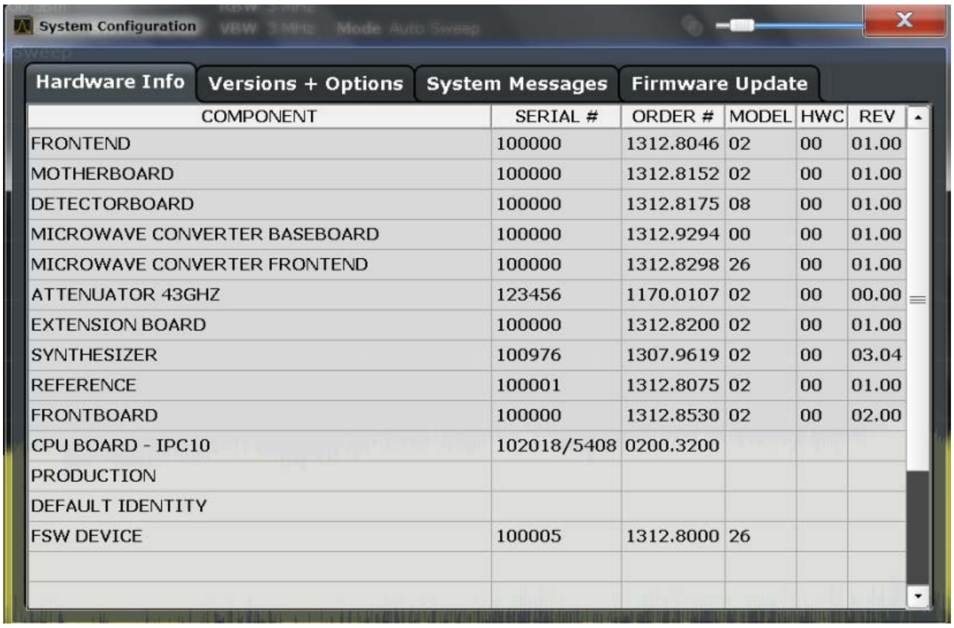
The system configuration information and settings are provided in the "System Configuration" dialog box which is displayed when you press the SETUP key and then select "System Configuration".

8.3.4.1 Hardware Information

An overview of the installed hardware in your R&S FSW is provided in the "Hardware Info" tab of the "System Configuration" dialog box.

Every listed component is described by its serial number, order number, model information, hardware code, and hardware revision.

This information can be useful when problems occur with the instrument and you require support from Rohde & Schwarz.



COMPONENT	SERIAL #	ORDER #	MODEL	HWC	REV
FRONTEND	100000	1312.8046	02	00	01.00
MOTHERBOARD	100000	1312.8152	02	00	01.00
DETECTORBOARD	100000	1312.8175	08	00	01.00
MICROWAVE CONVERTER BASEBOARD	100000	1312.9294	00	00	01.00
MICROWAVE CONVERTER FRONTEND	100000	1312.8298	26	00	01.00
ATTENUATOR 43GHZ	123456	1170.0107	02	00	00.00
EXTENSION BOARD	100000	1312.8200	02	00	01.00
SYNTHESIZER	100976	1307.9619	02	00	03.04
REFERENCE	100001	1312.8075	02	00	01.00
FRONTBOARD	100000	1312.8530	02	00	02.00
CPU BOARD - IPC10	102018/5408	0200.3200			
PRODUCTION					
DEFAULT IDENTITY					
FSW DEVICE	100005	1312.8000	26		

SCPI command:

DIAGnostic:SERVice:HWINfo? on page 718

8.3.4.2 Information on Versions and Options

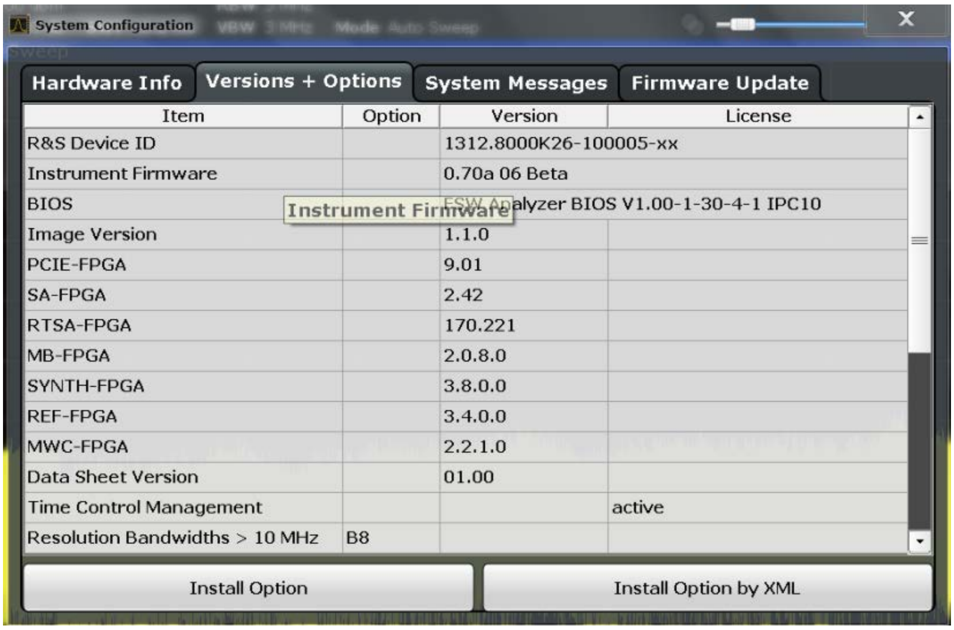
Information on the firmware version and options installed on your instrument is provided in the "Versions Options" tab of the "System Configuration" dialog box. The unique R&S device ID is also indicated here, as it is required for license and option administration.

You can also install new firmware options in this dialog box.

Expired option licenses

If an option is about to expire, a message box is displayed to inform you. You can then use the "Install Option" function to enter a new license key.

If an option has already expired, a message box appears for you to confirm. In this case, all instrument functions are unavailable (including remote control) until the R&S FSW is rebooted. You must then use the "Install Option" function to enter the new license key.



For details on options refer to the "Getting Started" manual, "Checking the Supplied Items".

SCPI commands:

`SYSTem:FORMat:IDENt` on page 720

`DIAGnostic:SERVice:BIOSinfo?` on page 718

Install Option

Opens an edit dialog box to enter the license key for the option that you want to install.

Only user accounts with administrator rights are able to install options.

Install Option by XML

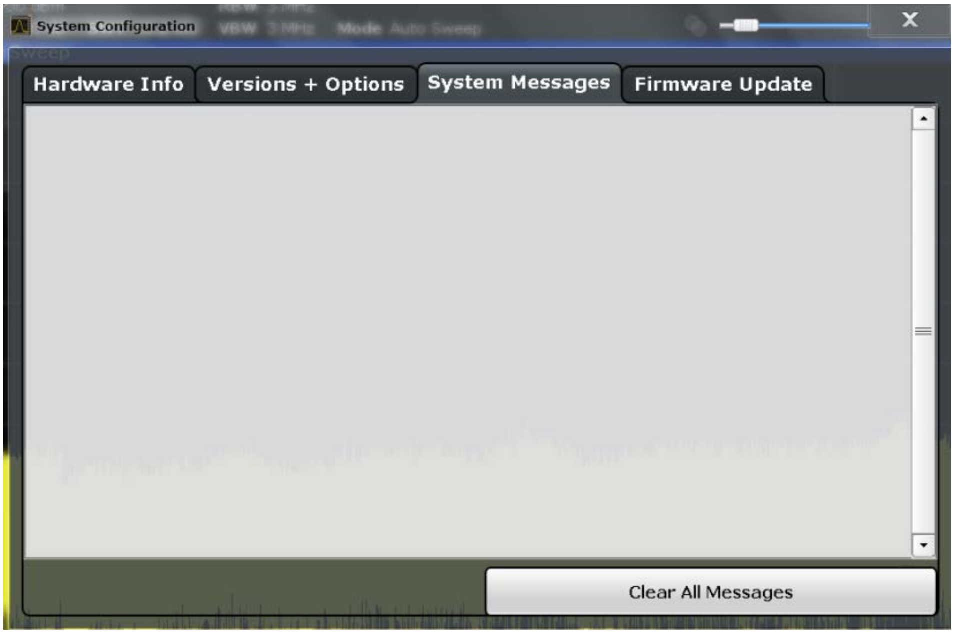
Opens a file selection dialog box to install an additional option to the R&S FSW using an XML file. Enter or browse for the name of an XML file that contains the option key and press "Select".

Only user accounts with administrator rights are able to install options.

8.3.4.3 System Messages

The system messages generated by the R&S FSW are displayed in the "System Messages" tab of the "System Configuration" dialog box.

The messages are displayed in the order of their occurrence; the most recent messages are placed at the top of the list. Messages that have occurred since you last visited the system messages tab are marked with an asterisk '*'.



If the number of error messages exceeds the capacity of the error buffer, "Message buffer overflow" is displayed. To clear the message buffer use the "Clear All Messages" button.

The following information is available:

No	device-specific error code
Message	brief description of the message
Component	hardware messages: name of the affected module
	software messages: name of the affected software
Date/Time	date and time of the occurrence of the message

SCPI command:

SYSTem:ERRor:LIST? on page 719

8.3.4.4 Firmware Updates

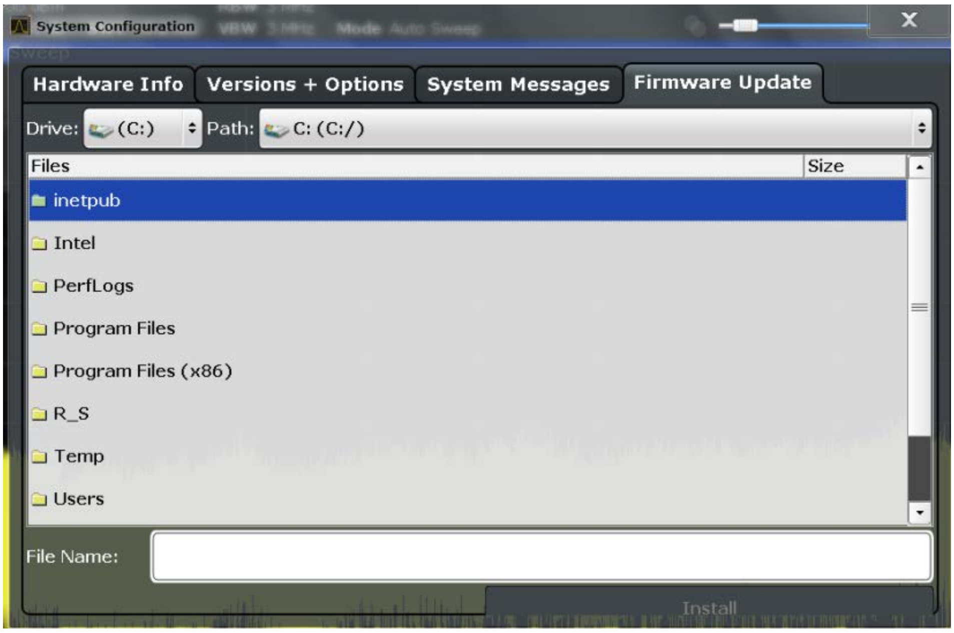
During instrument start, the installed hardware is checked against the current firmware version to ensure the hardware is supported. If not, an error message is displayed ("WRONG_FW") and you are asked to update the firmware. Until the firmware version is updated, self-alignment fails. To see which components are not supported, see the System Messages.

The firmware on your R&S FSW may also need to be updated in order to enable additional new features or if reasons for improvement come up. Ask your sales representative or check the Rohde&Schwarz website for availability of firmware updates. A firmware update package includes at least a setup file and release notes.

Before updating the firmware on your instrument, read the release notes delivered with the firmware version.

Only user accounts with administrator rights can perform a firmware update.

The firmware can be updated in the "Firmware Update" tab of the "System Configuration" dialog box.



Enter the name or browse for the firmware installation file and press the "Install" button.

8.3.4.5 Preset

The default operating mode is Signal and Spectrum Analyzer mode (SAN), however, the presetting can be changed to Multi-Standard Radio Analysis (MSRA). The defined operating mode is activated when you switch on the R&S FSW or press the PRESET key.

The presettings can be defined in the "Preset" tab of the "System Configuration" dialog box.

For details on operating modes see chapter 3, "Applications and Operating Modes", on page 18.

SCPI command:

`SYSTem:PRESet:COMPatible` on page 720

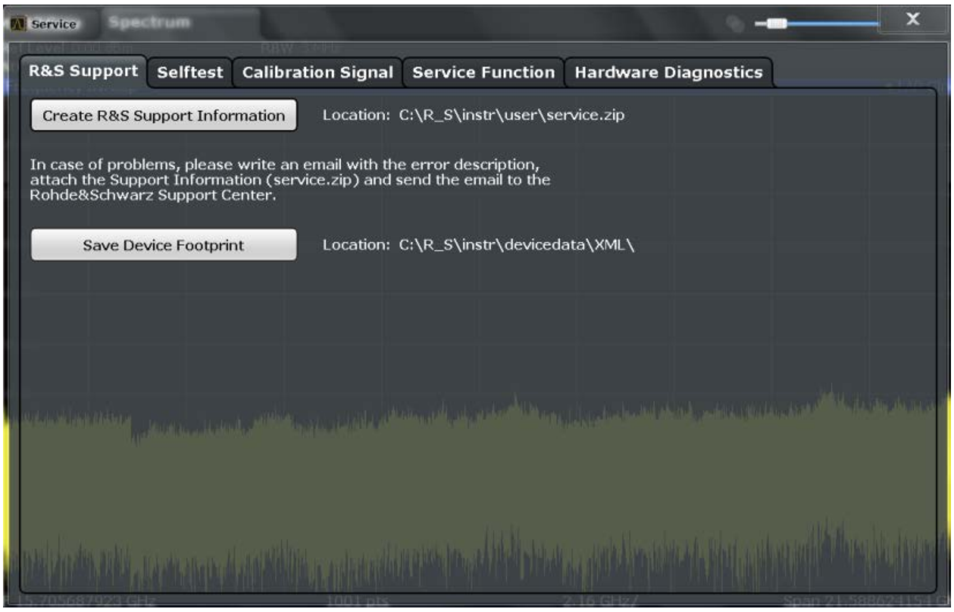
8.3.5 Service Functions

When unexpected problems arise with the R&S FSW some service functions may help you solve them.

The service functions are available in the "Service" dialog box which is displayed when you press the SETUP key and then select "Service".

8.3.5.1 R&S Support Information

In case of errors you can store useful information for troubleshooting and send it to your Rohde & Schwarz support center.



Creating R&S Support Information

Creates a *.zip file with important support information. The *.zip file contains the system configuration information ("device footprint"), the current eeprom data and a screenshot of the screen display.

This data is stored to the `C:\R_S\Instr\user\service.zip` file on the instrument.

If you contact the Rohde&Schwarz support to get help for a certain problem, send these files to the support in order to identify and solve the problem faster.

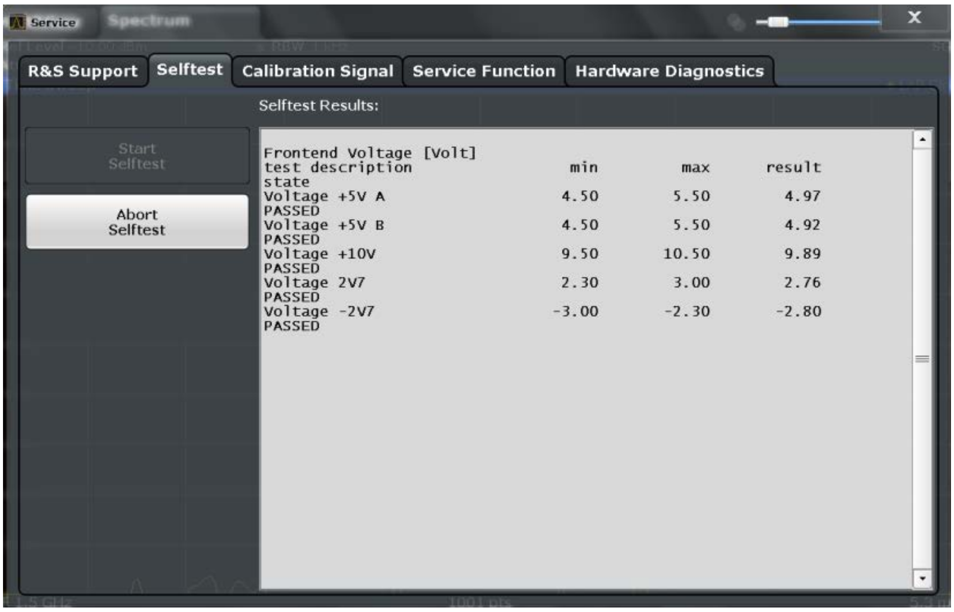
Save Device Footprint

Creates an *.xml file with information on installed hardware, software, image and FPGA versions. The *.xml file is stored under `C:\R_S\Instr\devicedata\xml\` on the instrument. It is also included in the `service.zip` file.

8.3.5.2 Selftest Settings and Results

If the R&S FSW fails you can perform a self test of the instrument to identify any defective modules.

The selftest settings and results are available in the "Selftest" tab of the "Service" dialog box.



Once the self test is started, all modules are checked consecutively and the test result is displayed. You can abort a running test.

In case of failure a short description of the failed test, the defective module, the associated value range and the corresponding test results are indicated.

A running Sequencer process is aborted when you start a self-alignment.

SCPI command:

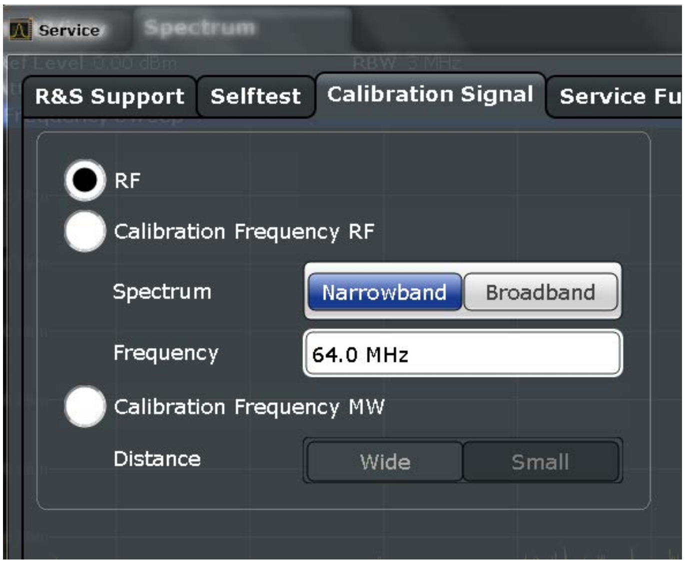
`*TST?` on page 449

`DIAGnostic:SERVice:STESt:RESult?` on page 704

8.3.5.3 Calibration Signal Display

Alternatively to the RF input signal from the front panel connector you can use the instrument's calibration signal as the input signal, for example to perform service functions on.

The calibration signal settings are available in the "Calibration Signal" tab of the "Service" dialog box.



RF

Uses the current RF signal at the input (default).

SCPI command:

`DIAGnostic:SERVice:INPut[:SELect]` on page 704

Calibration Frequency RF

Uses the internal calibration signal as the RF input signal.

SCPI command:

`DIAGnostic:SERVice:INPut:PULSed:CFRequency` on page 703

Spectrum ← Calibration Frequency RF

Defines whether a broadband or narrowband calibration signal is sent to the RF input.

"Narrowband" Used to calibrate the absolute level of the frontend at 64 MHz.

"Broadband" Used to calibrate the IF filter.

SCPI command:

`DIAGnostic:SERVice:INPut:RF:SPECtrum` on page 703

Frequency ← Calibration Frequency RF

Defines the frequency of the internal broadband calibration signal to be used for IF filter calibration (max. 64 MHz). For narrowband signals, 64 MHz is sent.

Calibration Frequency MW

Uses the microwave calibration signal as the RF input (for frequencies greater than 8 GHz; for R&S FSW 26 only). This function is used to calibrate the YIG-filter on the micro-wave converter.

The microwave calibration signal is pulsed. You can define whether the distance between input pulses is small or wide.

SCPI command:

DIAGnostic:SERVice:INPut:MC:DISTance on page 703

8.3.5.4 Service Functions

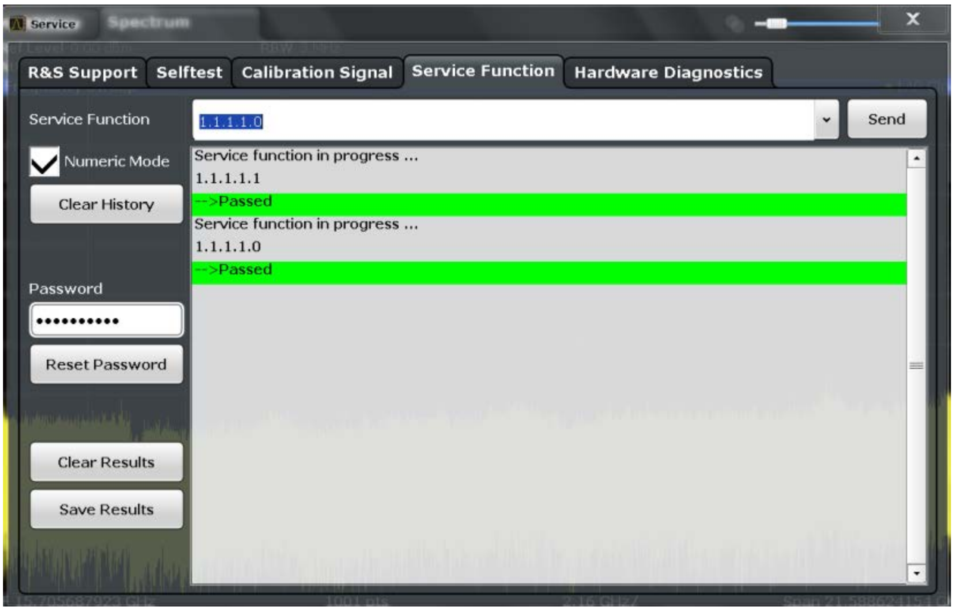
NOTICE

Using service functions

The service functions are not necessary for normal measurement operation. Incorrect use can affect correct operation and/or data integrity of the R&S FSW.

Therefore, only user accounts with administrator rights can use service functions and many of the functions can only be used after entering a password. These functions are described in the instrument service manual.

The service functions are available in the "Service Function" tab of the "Service" dialog box.



Service Function

Selects the service function by its numeric code or textual name.

The selection list includes all functions previously selected (since the last "Clear History" action).

SCPI command:
DIAGnostic:SERVice:SFUNction on page 720

Numeric Mode

If activated, the service function is selected by its numeric code. Otherwise, the function is selected by its textual name.

Send

Starts the selected service function.

SCPI command:
DIAGnostic:SERVice:SFUNction on page 720

Clear History

Deletes the list of previously selected service functions.

Password

Most service functions require a special password as they may disrupt normal operation of the R&S FSW. There are different levels of service functions, depending on how restrictive their use is handled. Each service level has a different password.

"Reset Password" returns to the lowest (least restrictive) service level.

Clear Results

Clears the result display for all previously performed service functions.

SCPI command:
DIAGnostic:SERVice:SFUNction:RESults:DELete on page 721

Save Results

Opens a file selection dialog box to save the results of all previously performed service functions to a file.

SCPI command:
DIAGnostic:SERVice:SFUNction:RESults:SAVE on page 721

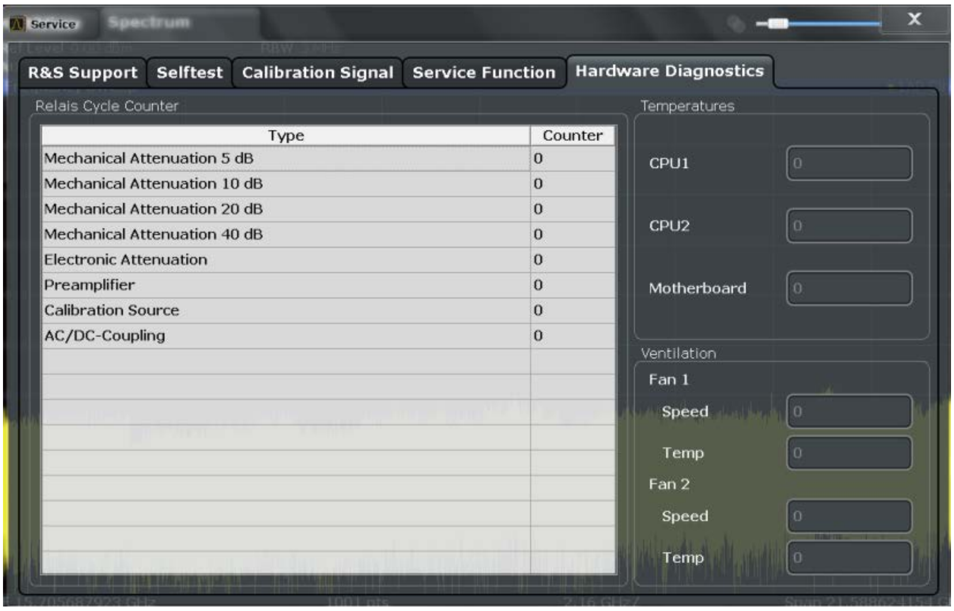
Result List

The Results List indicates the status and results of the executed service functions.

8.3.5.5 Hardware Diagnostics

In case problems occur with the instrument hardware, some diagnostic tools provide information that may support troubleshooting.

The hardware diagnostics tools are available in the "Hardware Diagnostics" tab of the "Service" dialog box.



Relay Cycle Counter
The hardware relays built into the R&S FSW may fail after a large number of switching cycles (see data sheet). The counter indicates how many switching cycles the individual relays have performed since they were installed.

Temperatures
Some hardware parts fail at high temperatures. Several temperature sensors in the R&S FSW provide the current temperature for the CPUs and the motherboard, which are indicated here.

Ventilation
High temperatures in the R&S FSW may occur when the fans fail. The current speed and temperatures of the built-in fans are displayed. High temperatures or very slow fan speed may indicate a hardware problem.

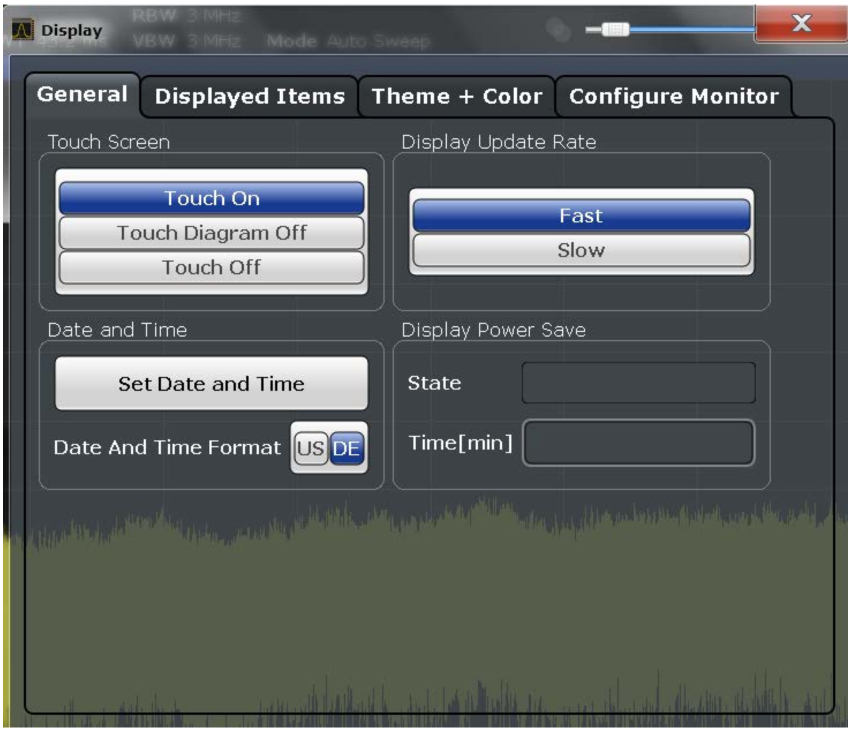
8.4 Display Settings

Some general display settings are available regardless of the current application or operating mode. For information on optimizing your display for measurement results see chapter 6.1, "Result Display Configuration", on page 228.

The general display settings are defined in the "Display" dialog box which is displayed when you press the SETUP key and then select "Display".

8.4.1 General Display Settings

This section includes general screen display behavior and date and time display. These settings are available in the "General" tab of the "Display" dialog box.



Deactivating and Activating the Touch Screen

The touch screen function can be deactivated, e.g. when the instrument is being used for demonstration purposes and tapping the screen should not provoke an action.

To reactivate the touch screen, simply press the SETUP key on the front panel. The "Display" dialog box is opened automatically and the "Touch Screen" option is set to "ON".

"TOUCH ON"	Touch screen function is active for the entire screen
"TOUCH OFF"	Touch screen is deactivated for the entire screen

"TOUCH DIAGRAM OFF"
Touch screen is deactivated for the diagram area of the screen, but active for the surrounding softkeys, toolbars and menus

SCPI command:
`DISPlay:TOUChscreen:STATe` on page 710

Display Update Rate
By default, a fast update rate ensures the most recent measurement results on the display. However, when performance is poor due to slow data transfer (for example during remote control), it may be helpful to decrease the frequency with which the screen display is updated.

Setting the Date and Time
The current date and time on the instrument is set using the standard Windows "Date and Time Properties" dialog box which is displayed when you select the "Set Date and Time" button in the "Display" dialog box, or when you tap the date and time display in the status bar.

Date and Time Format
Switches the time and date display on the screen between US and German (DE) format.

SCPI command:
`DISPlay[:WINDow]:TIME:FORMat` on page 710

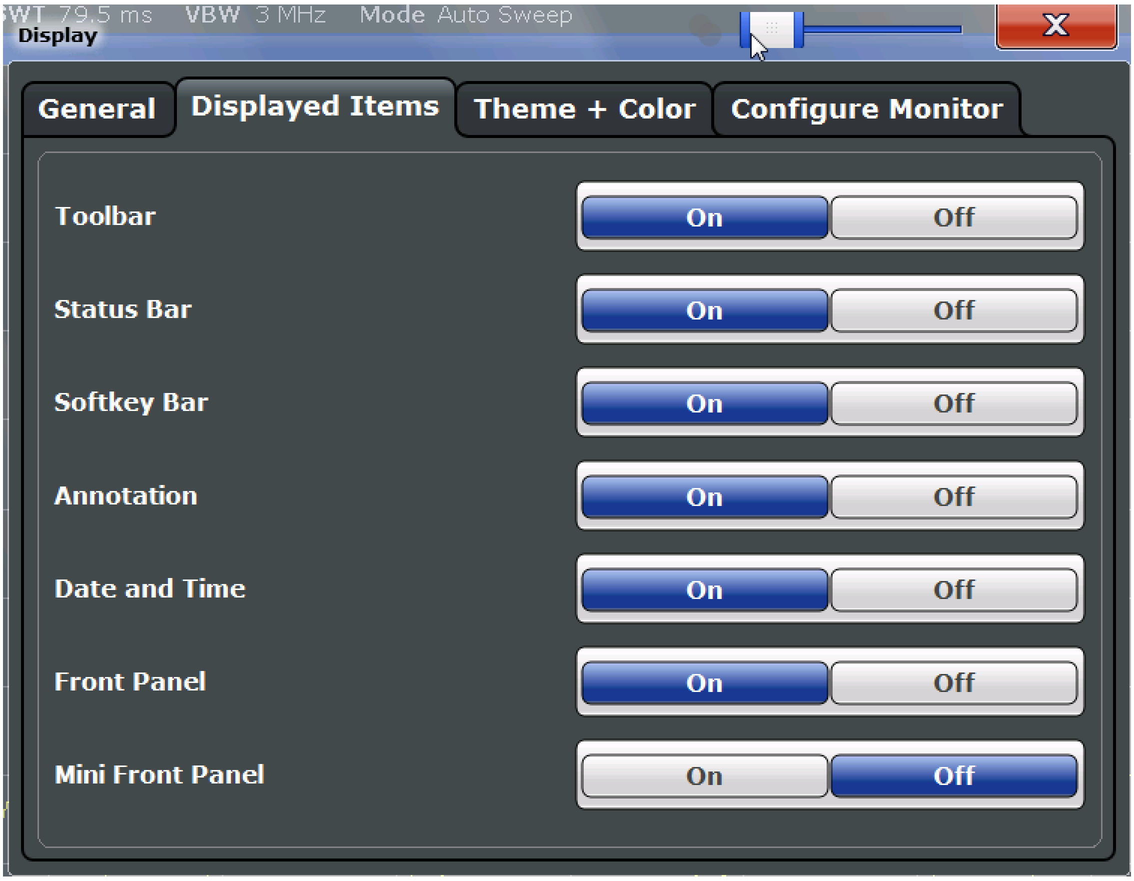
Display Power Save Function
The touch screen can be set to a power-save mode in which the display is temporarily switched off, including the backlight. This is useful during remote control, for example, or when a measurement with a long duration is running that needs not be monitored. You can define a waiting time after which the power-save mode sets in automatically if no manual interaction with the instrument occurs.

To switch the display back on, tap the screen or press a key.

SCPI command:
`DISPlay:PSAVe[:STATe]` on page 709
`DISPlay:PSAVe:HOLDoff` on page 708

8.4.2 Displayed Items

Several elements on the screen display can be hidden or shown as required, for example to enlarge the display area for the measurement results. These settings are available in the "Displayed Items" tab of the "Display" dialog box.



Toolbar

The toolbar provides access to frequently used functions via icons at the top of the screen. Some functions, such as zooming, finding help, printing screenshots or storing and loading files are not accessible at all without the toolbar.

SCPI command:

`DISPlay:TBAR[:STATe]` on page 709

Status Bar

The status bar beneath the diagram indicates the global instrument settings, the instrument status and any irregularities during measurement or display.

Some of the information displayed in the status bar can be queried from the status registry via remote commands, see chapter 10, "Remote Commands", on page 444.

SCPI command:

`DISPlay:SBAR[:STATe]` on page 709

Softkey Bar
Softkeys are virtual keys provided by the software. Thus, more functions can be provided than those that can be accessed directly via the function keys on the device.

The functions provided by the softkeys are often also available via dialog boxes. However, some functions may not be accessible at all without the softkey bar.

Note: The softkey bar is hidden while the SmartGrid is displayed and restored automatically when the SmartGrid is closed.

SCPI command:
`DISPlay:SKEYs[:STATe]` on page 709

Diagram Footer (Annotation)
The diagram footer beneath the diagram contains information on the x-axis of the diagram display, such as the current center frequency and span settings, the displayed span per division and the number of sweep points.

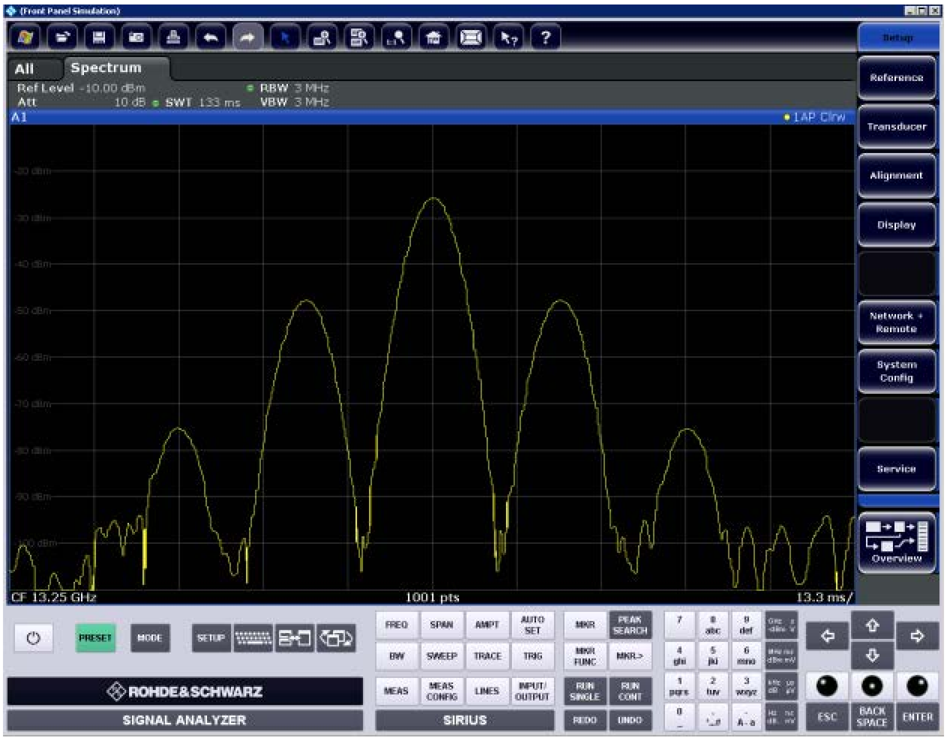
SCPI command:
`DISPlay:ANNotation:FREQuency` on page 708

Date and Time
The date and time display can be switched off independantly of the status bar.

You can set the current date and time and configure the display format in the "General" tab of the "Display" dialog box.

SCPI command:
`DISPlay[:WINDow]:TIME` on page 710

Front Panel
The "Front Panel" display simulates the entire front panel of the device (except for the external connectors) on the screen. This allows you to interact with the R&S FSW without requiring the keypad and keys located on the front panel of the device. This is useful, for example, when working with an external monitor or operating via remote control from a computer.



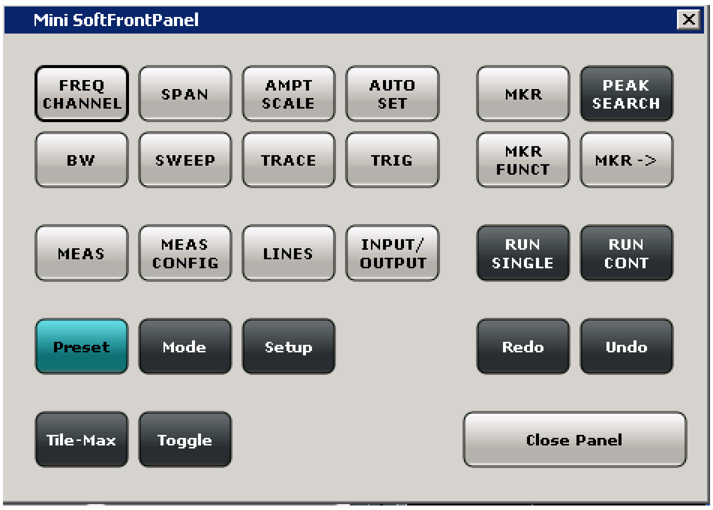
For more information see chapter 8.6.7, "How to Work with the Soft Front Panels", on page 379.

SCPI command:

`SYSTem:DISPlay:FPANel[:STATe]` on page 711

Mini Front Panel

If you require a front panel display but do not want to lose too much space for results in the display area, a mini front panel is available. The mini version displays only the main function hardkeys in a separate window in the display area. This window can be closed automatically after pressing a key ("Auto Close" option), or remain open, as desired.



Note:

You can also activate the Mini Front Panel using the key combination ALT + M. This is useful when you are working from a remote PC and the Front Panel function is not active.

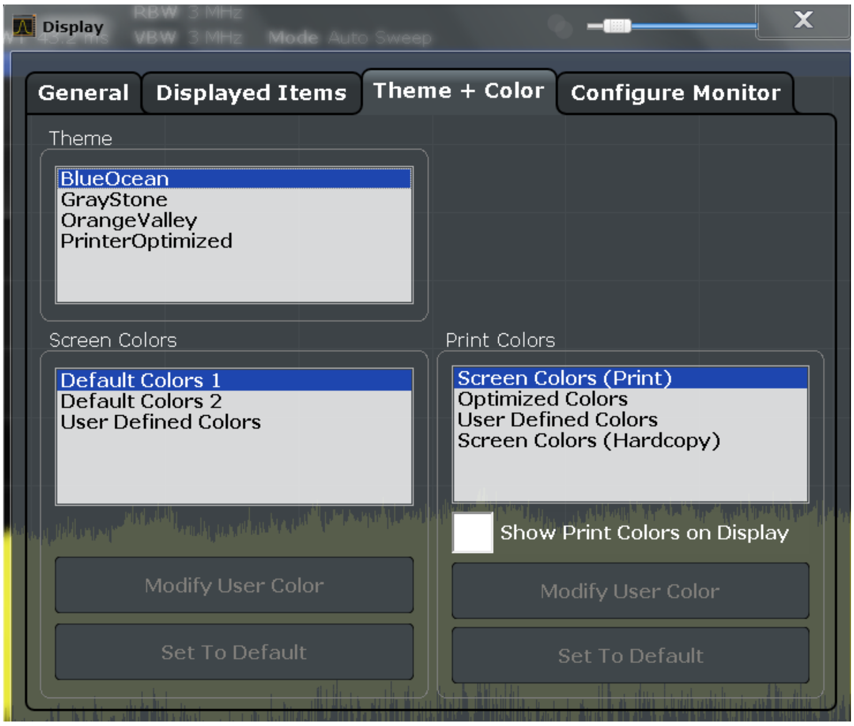
SCPI command:

`SYSTem:DISPlay:FPANel[:STATe]` on page 711

8.4.3 Display Theme and Colors

You can configure the used colors and styles of display elements on the screen. These settings are available in the "Theme + Color" tab of the "Display" dialog box.

For step-by-step instructions see chapter 8.6.6, "How to Configure the Colors for Display and Printing", on page 378.



Theme

The theme defines the colors and style used to display softkeys and other screen objects. The default theme is "BlueOcean".

SCPI command:

DISPlay:THEMe:SELect on page 713

Screen colors

Two different color sets are provided by the instrument, a third user-defined set can be configured.

The default color schemes provide optimum visibility of all screen objects when regarding the screen from above or below. Default setting is "Default Colors 1".

If "User Defined Colors" is selected, a user-defined color set can be defined.

SCPI command:

DISPlay:CMAP<item>:DEFault<colors> on page 711

Print colors

Defines the color settings used for printout. In addition to the predefined settings, a user-defined color set can be configured.

If "Show Print Colors on Display" is activated, the currently selected print colors are displayed as a preview for your selection.

Optimized Colors	Selects an optimized color setting for the printout to improve the visibility of the colors (default setting). Trace 1 is blue, trace 2 black, trace 3 green, and the markers are turquoise. The background is always printed in white and the grid in black.
Screen Colors (Print)	Selects the current screen colors for the printout. The background is always printed in white and the grid in black.
Screen Colors (Hardcopy)	Selects the current screen colors without any changes for a hardcopy.
User Defined Colors	Selects the user-defined color setting.

SCPI command:

HCOPy:CMAP<item>:DEFault<colors> on page 691

Modifying User-Defined Colors

You can configure the colors used to display and print individual screen objects according to your specific requirements.

The colors are configured in the (identical) "Screen Color Setup"/"Printer Color Setup" dialog boxes.

Selecting the Object ← Modifying User-Defined Colors
Selects the object for which the color is to be defined. Colors can be defined for the following objects:

- Background
- Grid
- Individual traces
- Display lines
- Limit lines and check results
- Markers and marker information

SCPI command:
Each object is assigned to a specific suffix of the CMAP commands, see chapter 10.8.5.3, "CMAP Suffix Assignment", on page 713.

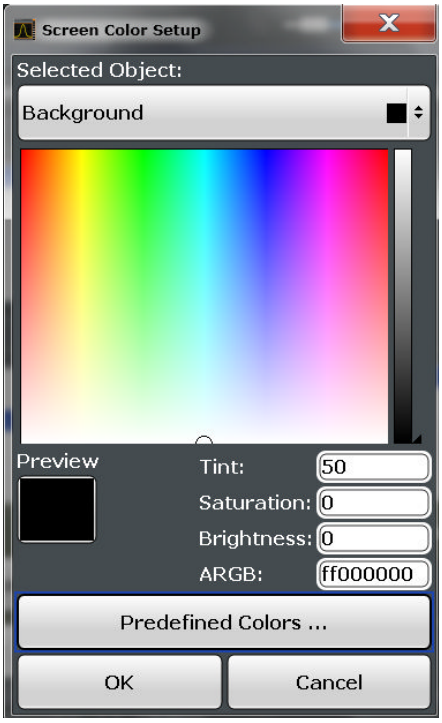
Predefined Colors ← Modifying User-Defined Colors
Displays the available colors from the predefined color set that can be used for the selected object.

SCPI command:
`HCOPy:CMAP<item>:PDEFined` on page 692

Preview
Indicates the currently selected color that will be used for the selected object.

Defining User-specific Colors
In addition to the colors in the predefined color set you can configure a user-specific color to be used for the selected object.

When you select "Userdefined Colors", the set of predefined colors is replaced by a color palette and color configuration settings.



The color palette allows you to select the color directly. The color settings allow you to define values for tint, saturation and brightness.

SCPI command:

`HCOPy:CMAP<item>:HSL` on page 691

Restoring the User Settings to Default Colors

In addition to the predefined color settings, a user-defined setting can be configured. By default, the same settings as defined in "Default Colors 1" are used. They can then be modified according to user-specific requirements (see "Modifying User-Defined Colors" on page 370).

The "Set to Default" function restores the original default settings for the user-defined color set. You can select which of the three default settings are restored.

SCPI command:

`DISPlay:CMAP<item>:PDEFined` on page 712

8.5 External Monitor Settings

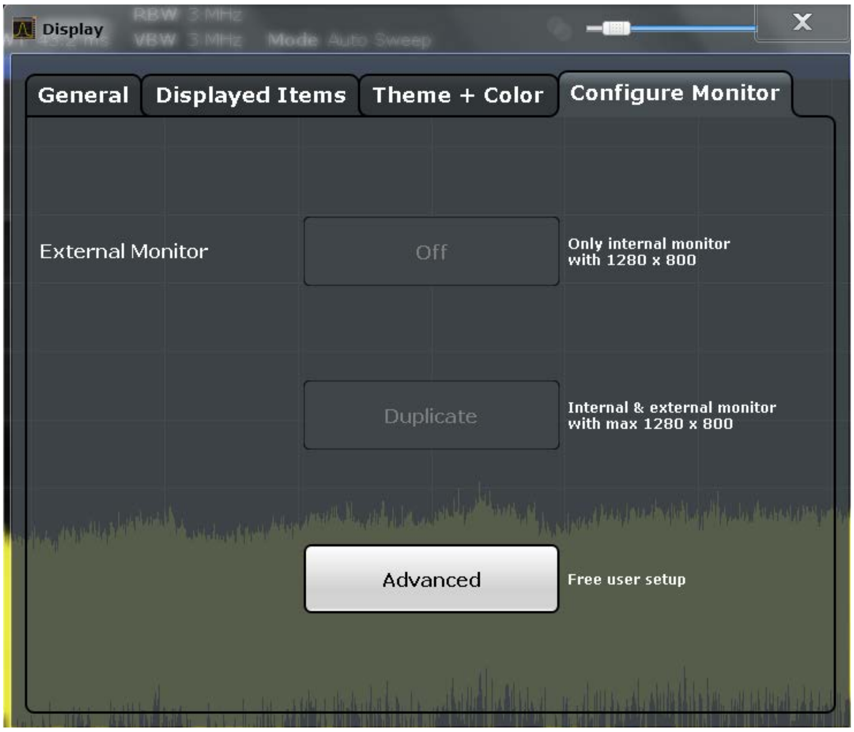
You can connect an external monitor (or projector) to the DVI or DISPLAY PORT connector on the instrument's rear panel (see the R&S FSW Getting Started manual).

Which display device is used by the instrument is configured in the "Configure Monitor" tab of the "Display" dialog box.

Screen resolution and format

The touch screen of the R&S FSW is calibrated for a 16:10 format. If you connect a monitor or projector using a different format (e.g. 4:3), the calibration will not be correct and the screen will not react to your touch actions properly.

The touch screen has a screen resolution of 1280x800 pixels. Most external monitors have a higher screen resolution. If the screen resolution of the monitor is set higher than the instrument's resolution, the application window uses an area of 1280x800 pixels on the monitor display. For full screen display, adjust the monitor's screen resolution.



External Monitor Off
Only the internal monitor of the R&S FSW is used for display.

Duplicate
Both the internal and the external monitor are used.

Advanced
User-defined configuration; opens the standard Windows configuration dialog box to configure the display devices to be used

8.6 How to Configure the Basic Instrument Settings

The following step-by-step instructions demonstrate how to configure the basic instrument settings. For details on individual functions and settings see chapter 8.3, "General Instrument Settings", on page 343.

The remote commands required to perform these tasks are described in chapter 8.3, "General Instrument Settings", on page 343.

8.6.1 How to Perform a Self Test

The self test does not need to be repeated every time the instrument is switched on. It is only necessary when instrument malfunction is suspected.

Operating temperature

Before performing this functional test, make sure that the instrument has reached its operating temperature (for details, refer to the data sheet).

1. Press the SETUP key.
2. Press the "Service" softkey.
3. Press the "Selftest" softkey.

   Once the instrument modules have been checked successfully, a message is displayed.

8.6.2 How to Align the Instrument and the Touch Screen

Operating temperature

Before performing this functional test, make sure that the instrument has reached its operating temperature (for details, refer to the data sheet).

To perform a self-alignment

1. Press the SETUP key.
2. Select the "Alignment" softkey.
3. Select the "Start Self-alignment" button.
4. To abort the self-alignment process, select the "Abort Self-alignment" button.

   Once the system correction values have been calculated successfully, a message is displayed.

To display the alignment results again later

- Press the SETUP key.
- Press the "Alignment" softkey.

To align the touch screen

1. Press the SETUP key.
2. Select the "Alignment" softkey.
3. Select "Touch Screen Alignment".

   A blinking cross appears in the lower left corner of the screen.
4. Touch and hold the blinking cross until it stops blinking.
   Repeat this action for the crosses in the other corners.

8.6.3 How to Install an R&S FSW Option

Additional options for the R&S FSW can be enabled using a license key. To obtain the license key, consult your sales representative. You need the device ID and serial number of your instrument to get a license key (see chapter 8.3.4, "System Configuration Settings", on page 353). No additional installation is required.

8.6.4 How to Update the Instrument Firmware

Only user accounts with administrator rights can perform a firmware update.

1. Download the update package from the Rohde&Schwarz website and store it on a memory stick, on the instrument, or on a server network drive that can be accessed by the instrument.
2. **NOTICE!** Stop measurement. The firmware update must not be performed during a running measurement.

   If a measurement is running, stop it by pressing the highlighted RUN CONT or RUN SINGLE key.
3. Press the SETUP key.
4. Select the "Firmware Update" tab.
5. In the file selection dialog box select the `FSWSetup*.exe` file.
6. Tap "Install" to start the update.
7. After the firmware update, the R&S FSW reboots automatically.
8. Depending on the previous firmware version, a reconfiguration of the hardware might be required during the first startup of the firmware. The reconfiguration starts auto-

matically, and a message box informs you about the process. When the reconfiguration has finished, the instrument again reboots automatically.

Note: Do not switch off the instrument during the reconfiguration process!

Now the firmware update is complete. It is recommended that you perform a self-alignment after the update (see chapter 8.6.2, "How to Align the Instrument and the Touch Screen", on page 374).

8.6.5 How to Configure the Transducer

Configuring the transducer is very similar to configuring limit lines.

The transducer settings are defined in the "Transducer" dialog box which is displayed when you press the SETUP key and then select "Transducer".

The following tasks are described:

- "How to find compatible transducer lines" on page 376
- "How to activate and deactivate a transducer" on page 376
- "How to edit existing transducer lines" on page 376
- "How to copy an existing transducer line" on page 377
- "How to delete an existing transducer line" on page 377
- "How to configure a new transducer line" on page 377
- "How to move the transducer line vertically or horizontally" on page 378

How to find compatible transducer lines

► In the "Transducer" dialog box, select the "View filter" option: "Show compatible".

All transducer lines stored on the instrument that are compatible to the current measurement settings are displayed in the overview.

How to activate and deactivate a transducer

1. To activate a transducer select a transducer line in the overview and select the "Active" setting for it.

   The trace is automatically recalculated for the next sweep after a transducer line is activated.

2. To deactivate a transducer line, deactivate the "Active" setting for it.

   After the next sweep, the originally measured values are displayed.

How to edit existing transducer lines

Existing transducer line configurations can be edited.

1. In the "Transducer" dialog box, select the transducer line.

2. Select the "Edit" button.

3. Edit the line configuration as described in "How to configure a new transducer line" on page 377.
4. Save the new configuration by selecting the "Save" button.

   The trace is automatically recalculated for the next sweep if the transducer line is active.

How to copy an existing transducer line

1. In the "Transducer" dialog box, select the transducer line.
2. Select the "Copy" button.

   The "Edit Transducer" dialog box is opened with the configuration of the selected transducer.
3. Define a new name to create a new transducer with the same configuration as the source line.
4. Edit the line configuration as described in "How to configure a new transducer line" on page 377.
5. Save the new configuration by selecting the "Save" button.

   The new transducer line is displayed in the overview and can be activated.

How to delete an existing transducer line

1. In the "Transducer" dialog box, select the transducer line.
2. Select the "Delete" button.
3. Confirm the message.

   The transducer line is deleted. After the next sweep, the originally measured values are displayed.

How to configure a new transducer line

1. In the "Transducer" dialog box, select the "New" button.

   The "Edit Transducer" dialog box is displayed. The current line configuration is displayed in the preview area of the dialog box. The preview is updated after each change to the configuration.
2. Define a "Name" and, optionally, a "Comment" for the new transducer line.
3. Define the scaling for the x-axis.
4. Define the data points: minimum 2, maximum 50:
   a) Select "Insert Value".
   b) Define the x-value ("Position") and y-value ("Value") of the first data point.
   c) Select "Insert Value" again and define the second data point.

d) Repeat this to insert all other data points.
To insert a data point before an existing one, select the data point and then "Insert Value".
To insert a new data point at the end of the list, move the focus to the line after the last entry and then select "Insert Value".
To delete a data point, select the entry and then "Delete Value".

5. Check the current line configuration in the preview area of the dialog box. If necessary, correct individual data points or add or delete some.
If necessary, shift the entire line vertically or horizontally by selecting the "Shift x" or "Shift y" button and defining the shift width.
6. Save the new configuration by selecting the "Save" button.

   The new transducer line is displayed in the overview and can be activated.

How to move the transducer line vertically or horizontally

A configured transducer line can easily be moved vertically or horizontally. Thus, a new transducer line can be easily generated based upon an existing transducer line which has been shifted.

1. In the "Line Config" dialog box, select the transducer line.
2. Select the "Edit" button.
3. In the "Edit transducer Line" dialog box, select the "Shift x" or "Shift y" button and define the shift width.
4. Save the shifted data points by selecting the "Save" button.

   If activated, the trace is recalculated after the next sweep.

8.6.6 How to Configure the Colors for Display and Printing

You can configure the style and colors with which various screen objects are displayed or printed.

To select a color set

1. Press the SETUP key and select the "Display" softkey.
2. Select the "Theme + Color" tab.
3. In the "Screen Colors" area, select a predefined set of colors to be used for screen display, or select "User Defined Colors" to configure the color set yourself.
4. In the "Print Colors" area, select a predefined set of colors to be used for printing screenshots, or select "User Defined Colors" to configure the color set yourself.
Activate the "Show Print Colors on Display" option to see a preview of the print colors.

To configure a user-defined color set

1. In the "Theme + Color" tab of the "Display" dialog box select "User Defined Colors" either for the screen or the print colors.
2. Select "Modify User Color".

   The "Screen Color Setup" dialog box is opened.
3. From the "Selected Object" list, select the object to which you want to assign a color.
4. Select a color from the "Predefined Colors" or select the "Userdefined Colors..." button to define a different color.

   The "Preview" area indicates the currently selected color.
5. To assign a user-specific color to the selected object, do one of the following:
   - Select the color from the palette.
   - Enter values for the "Tint", "Saturation", and "Brightness".
     Note: In the continuous color spectrum ("Tint") red is represented by 0% and blue by 100%.
   - Enter an "ARGB" value in hexadecimal format.
6. Select the next object to which you want to assign a color from the "Selected Object" list and assign a color as described.
   Repeat these steps until all objects you want to configure have been assigned a color.
7. Select "OK" to close the dialog box and apply the colors to the assigned objects.

8.6.7 How to Work with the Soft Front Panels

Basic operation with the soft front panels is identical to normal operation, except for the following aspects:

To activate a key, select the key on the touch screen.

To simulate the use of the rotary knob, use the additional keys displayed between the keypad and the arrow keys:

Icon	Function
	Turn left
	Enter
	Turn right

Mini Front Panel

The Mini Front Panel provides only the hardkeys on the touchscreen, in order to operate the R&S FSW via an external monitor or remote desktop.

By default, the "Auto close" option is activated and the Mini Front Panel window closes automatically after you select a key. This is useful if you only require the Mini Front Panel display occassionally to press a single function key.

If you want the window to remain open, deactivate the "Auto close" option. You can close the window manually by selecting "Close Panel" or the key combination ALT + M.

9 Network and Remote Operation

In addition to working with the R&S FSW interactively, located directly at the instrument, it is also possible to operate and control it from a remote PC. Various methods for remote control are supported:

- Connecting the instrument to a (LAN) network
- Using the LXI browser interface in a LAN network
- Using the Windows Remote Desktop application in a LAN network
- Connecting a PC via the GPIB interface

How to configure the remote control interfaces is described in chapter 9.5, "How to Set Up a Network and Remote Control", on page 425.

9.1 Remote Control Basics

Basic information on operating an instrument via remote control is provided here. this information applies to all applications and operating modes on the R&S FSW.

9.1.1 Remote Control Interfaces and Protocols

The instrument supports different interfaces for remote control. The following table gives an overview.

Table 9-1: Remote control interfaces and protocols

Interface	Protocols, VISA*) address string	Remarks

Local Area Network (LAN)	Protocols:
	VXI-11, HiSLIP
	VISA*) address string:
	`TCPIP::host_address[::LAN_device_name][::INSTR]`
	Raw socket
	VISA*) address string:
	`TCPIP::host_address::port::SOCKET`
		A LAN connector is located on the rear panel of the instrument.
		The interface is based on TCP/IP and supports various protocols.
		For a description of the protocols refer to:
		VXI-11 Protocol
		"HiSLIP Protocol" on page 384
		"Socket Communication" on page 385

GPIB (IEC/ IEEE Bus Interface)	VISA*) address string: `GPIB::primary address[::INSTR]` (no secondary address)	A GPIB bus interface (option R&S FSW-B10) according to the IEC 625.1/ IEEE 488.1 standard is located on the rear panel of the instrument. For a description of the interface refer to 9.1.1.2 GPIB Interface (IEC 625/ IEEE 418 Bus Interface).
USB	VISA*) address string: `USB::<vendor ID>::<product_ID>::<serial_number>[::INSTR]`	USB connectors are located on the rear panel of the instrument. For a description of the interface refer to 9.1.1.3 USB Interface.

*) VISA is a standardized software interface library providing input and output functions to communicate with instruments. A VISA installation on the controller is a prerequisite for remote control using the indicated interfaces (see also chapter 9.1.3, "VISA Libraries", on page 386).

Within this interface description, the term GPIB is used as a synonym for the IEC/IEEE bus interface.

9.1.1.1 LAN Interface

To be integrated in a LAN, the instrument is equipped with a LAN interface, consisting of a connector, a network interface card and protocols. The network card can be operated with the following interfaces:

- 10 Mbit/s Ethernet IEEE 802.3
- 100 Mbit/s Ethernet IEEE 802.3u
- 1Gbit/s Ethernet IEEE 802.3ab

For remote control via a network, the PC and the instrument must be connected via the LAN interface to a common network with TCP/IP network protocol. They are connected using a commercial RJ45 cable (shielded or unshielded twisted pair category 5). The TCP/IP network protocol and the associated network services are preconfigured on the instrument. Software for instrument control and the VISA program library must be installed on the controller.

VISA library

Instrument access is usually achieved from high level programming platforms using VISA as an intermediate abstraction layer. VISA encapsulates the low level VXI, GPIB, LAN or USB function calls and thus makes the transport interface transparent for the user. See chapter 9.1.3, "VISA Libraries", on page 386 for details.

The R&S FSW supports various LAN protocols such as LXI, RSIB, raw socket or the newer HiSLIP protocol.

IP address

Only the IP address or a valid DNS host name is required to set up the connection. The host address is part of the "VISA resource string" used by the programs to identify and control the instrument.

The VISA resource string has the form:

`TCPIP::host address[::LAN device name][::INSTR]`

or

`TCPIP::host address::port::SOCKET`

where:

- **TCPIP** designates the network protocol used
- **host address** is the IP address or host name of the device
- **LAN device name** defines the protocol and the instance number of a sub-instrument;
  - `inst0` selects the VXI-11 protocol (default)
  - `hislip0` selects the newer HiSLIP protocol
- **INSTR** indicates the instrument resource class (optional)
- **port** determines the used port number
- **SOCKET** indicates the raw network socket resource class

Example:

- Instrument has the IP address *192.1.2.3*; the valid resource string using VXI-11 protocol is:
  `TCPIP::192.1.2.3::INSTR`
- The DNS host name name is *FSW-123456*; the valid resource string using HiSLIP is:
  `TCPIP::FSW-123456::hislip0`
- A raw socket connection can be established using:
  `TCPIP::192.1.2.3::5025::SOCKET`

Identifying instruments in a network

If several instruments are connected to the network, each instrument has its own IP address and associated resource string. The controller identifies these instruments by means of the resource string.

For details on configuring the LAN connection, see chapter 9.5.1, "How to Configure a Network", on page 426.

VXI-11 Protocol

The VXI-11 standard is based on the ONC RPC (Open Network Computing Remote Procedure Call) protocol which in turn relies on TCP/IP as the network/transport layer. The TCP/IP network protocol and the associated network services are preconfigured. TCP/IP ensures connection-oriented communication, where the order of the exchanged messages is adhered to and interrupted links are identified. With this protocol, messages cannot be lost.

HiSLIP Protocol

The HiSLIP (**Hi**gh **S**peed **L**AN **I**nstrument **P**rotocol) is the successor protocol for VXI-11 for TCP-based instruments specified by the IVI foundation. The protocol uses two TCP sockets for a single connection - one for fast data transfer, the other for non-sequential control commands (e.g. `Device Clear` or `SRQ`).

HiSLIP has the following characteristics:

- High performance as with raw socket network connections
- Compatible IEEE 488.2 support for Message Exchange Protocol, Device Clear, Serial Poll, Remote/Local, Trigger, and Service Request
- Uses a single IANA registered port (4880), which simplifies the configuration of firewalls
- Supports simultaneous access of multiple users by providing versatile locking mechanisms
- Usable for IPv6 or IPv4 networks

Note that HiSLIP data is sent to the device using the "fire and forget" method with immediate return, as opposed to VXI-11, where each operation is blocked until a VXI-11 device handshake returns. Thus, a successful return of a VISA operation such as `viWrite()` does not guarantee that the instrument has finished or started the requested command, but is delivered to the TCP/IP buffers.

Socket Communication

An alternative way for remote control of the software is to establish a simple network communication using sockets. The socket communication, also referred as “Raw Ethernet communication”, does not necessary require a VISA installation on the remote controller side.

The simplest way to establish socket communication is to use the built-in telnet program. The telnet program is part of every operating system and supports a communication with the software on a command-by-command basis. For better utilization and to enable automation by means of programs, user defined sockets can be programmed.

Socket connections are established on a specially defined port. The socket address is a combination of the IP address or the host name of the instrument and the number of the port configured for remote-control. All Signal and Spectrum Analyzers use port number 5025 for this purpose. The port is configured for communication on a command-to-command basis and for remote control from a program.

9.1.1.2 GPIB Interface (IEC 625/IEEE 418 Bus Interface)

A GPIB interface is integrated on the rear panel of the instrument. By connecting a PC to the R&S FSW via the GPIB connection you can send remote commands to control and operate the instrument.

To be able to control the instrument via the GPIB bus, the instrument and the controller must be linked by a GPIB bus cable. A GPIB bus card, the card drivers and the program libraries for the programming language used must be provided in the controller. The controller must address the instrument with the GPIB bus address (see chapter 9.5.1.5, "How to Change the GPIB Instrument Address", on page 432). You can set the GPIB address and the ID response string. The GPIB language is set as SCPI by default and cannot be changed for the R&S FSW.

Notes and Conditions

In connection with the GPIB interface, note the following:

- Up to 15 instruments can be connected
- The total cable length is restricted to a maximum of 15 m or 2 m times the number of devices, whichever is less; the cable lenth between two instruments should not exceed 2 m.
- A wired "OR"-connection is used if several instruments are connected in parallel.
- Any connected IEC-bus cables should be terminated by an instrument or controller.

9.1.1.3 USB Interface

For remote control via the USB connection, the PC and the instrument must be connected via the USB type B interface. A USB connection requires the VISA library to be installed. VISA detects and configures the R&S instrument automatically when the USB connection is established. You do not have to enter an address string or install a separate driver.

USB address

The used USB address string is:

`USB::<vendor ID>::<product ID>::<serial number>[::INSTR]`

where:

- <vendor ID> is the vendor ID for Rohde&Schwarz
- <product ID> is the product ID for the R&S instrument
- <serial number> is the individual serial number on the rear of the instrument

Example:

`USB::0x0AAD::0x00C6::100001::INSTR`

0x0AAD is the vendor ID for Rohde&Schwarz

0xC6 is the product ID for the R&S FSW8

100001 is the serial number of the particular instrument

9.1.2 SCPI (Standard Commands for Programmable Instruments)

SCPI commands - messages - are used for remote control. Commands that are not taken from the SCPI standard follow the SCPI syntax rules. The instrument supports the SCPI version 1999. The SCPI standard is based on standard IEEE 488.2 and aims at the standardization of device-specific commands, error handling and the status registers. The tutorial "Automatic Measurement Control - A tutorial on SCPI and IEEE 488.2" from John M. Pieper (R&S order number 0002.3536.00) offers detailed information on concepts and definitions of SCPI.

Tables provide a fast overview of the bit assignment in the status registers. The tables are supplemented by a comprehensive description of the status registers.

9.1.3 VISA Libraries

VISA is a standardized software interface library providing input and output functions to communicate with instruments. The I/O channel (LAN or TCP/IP, USB, GPIB,...) is selected at initialization time by means of the channel–specific address string ("VISA resource string") indicated in table 9-1, or by an appropriately defined VISA alias (short name).

A VISA installation is a prerequisite for remote control using the following interfaces:

- chapter 9.1.1.1, "LAN Interface", on page 382

- chapter 9.1.1.2, "GPIB Interface (IEC 625/IEEE 418 Bus Interface)", on page 385
- chapter 9.1.1.3, "USB Interface", on page 386

For more information about VISA refer to the user documentation.

9.1.4 Messages

The messages transferred on the data lines are divided into the following categories:

- Interface messages
  Interface messages are transmitted to the instrument on the data lines, with the attention line being active (LOW). They are used to communicate between the controller and the instrument. Interface messages can only be sent by instruments that have GPIB bus functionality. For details see the sections for the required interface.
- Instrument messages
  Instrument messages are employed in the same way for all interfaces, if not indicated otherwise in the description. Structure and syntax of the instrument messages are described in chapter 9.1.5, "SCPI Command Structure", on page 388. A detailed description of all messages available for the instrument is provided in the chapter "Remote Control Commands".
  There are different types of instrument messages, depending on the direction they are sent:
  – Commands
  – Instrument responses

Commands

Commands (program messages) are messages the controller sends to the instrument. They operate the instrument functions and request information. The commands are subdivided according to two criteria:

- According to the effect they have on the instrument:
  – **Setting commands** cause instrument settings such as a reset of the instrument or setting the frequency.
  – **Queries** cause data to be provided for remote control, e.g. for identification of the instrument or polling a parameter value. Queries are formed by directly appending a question mark to the command header.
- According to their definition in standards:
  – **Common commands**: their function and syntax are precisely defined in standard IEEE 488.2. They are employed identically on all instruments (if implemented). They refer to functions such as management of the standardized status registers, reset and self test.
  – **Instrument control commands** refer to functions depending on the features of the instrument such as frequency settings. Many of these commands have also been standardized by the SCPI committee. These commands are marked as "SCPI compliant" in the command reference chapters. Commands without this SCPI label are device-specific, however, their syntax follows SCPI rules as permitted by the standard.

Instrument responses

Instrument responses (response messages and service requests) are messages the instrument sends to the controller after a query. They can contain measurement results, instrument settings and information on the instrument status.

9.1.5 SCPI Command Structure

SCPI commands consist of a so-called header and, in most cases, one or more parameters. The header and the parameters are separated by a "white space" (ASCII code 0 to 9, 11 to 32 decimal, e.g. blank). The headers may consist of several mnemonics (keywords). Queries are formed by appending a question mark directly to the header.

The commands can be either device-specific or device-independent (common commands). Common and device-specific commands differ in their syntax.

9.1.5.1 Syntax for Common Commands

Common (=device-independent) commands consist of a header preceded by an asterisk (*) and possibly one or more parameters.

Examples:

*RST	RESET	Resets the instrument.
*ESE	EVENT STATUS ENABLE	Sets the bits of the event status enable registers.
*ESR?	EVENT STATUS QUERY	Queries the contents of the event status register.
*IDN?	IDENTIFICATION QUERY	Queries the instrument identification string.

9.1.5.2 Syntax for Device-Specific Commands

Not all commands used in the following examples are necessarily implemented in the instrument.

For demonstration purposes only, assume the existence of the following commands for this section:

- `DISPlay[:WINDow<1...4>]:MAXimize <Boolean>`
- `FORMat:READings:DATA <type>[,<length>]`
- `HCOPy:DEVice:COLor <Boolean>`
- `HCOPy:DEVice:CMAP:COLor:RGB <red>,<green>,<blue>`
- `HCOPy[:IMMediate]`
- `HCOPy:ITEM:ALL`
- `HCOPy:ITEM:LABel <string>`
- `HCOPy:PAGE:DIMensions:QUADrant[<N>]`
- `HCOPy:PAGE:ORIentation LANDscape | PORTrait`
- `HCOPy:PAGE:SCALe <numeric value>`
- `MMEMory:COPY <file_source>,<file_destination>`
- `SENSE:BANDwidth|BWIDth[:RESolution] <numeric_value>`
- `SENSe:FREQuency:STOP <numeric value>`
- `SENSe:LIST:FREQuency <numeric_value>{,<numeric_value>}`

Long and short form

The mnemonics feature a long form and a short form. The short form is marked by upper case letters, the long form corresponds to the complete word. Either the short form or the long form can be entered; other abbreviations are not permitted.

Example:

`HCOPy:DEVice:COLor ON` is equivalent to `HCOP:DEV:COL ON`.

Case-insensitivity

Upper case and lower case notation only serves to distinguish the two forms in the manual, the instrument itself is case-insensitive.

Numeric suffixes

If a command can be applied to multiple instances of an object, e.g. specific channels or sources, the required instances can be specified by a suffix added to the command. Numeric suffixes are indicated by angular brackets (<1...4>, <n>, <i>) and are replaced by a single value in the command. Entries without a suffix are interpreted as having the suffix 1.

Example:

Definition: `HCOPy:PAGE:DIMensions:QUADrant[<N>]`

Command: `HCOP:PAGE:DIM:QUAD2`

This command refers to the quadrant 2.

Different numbering in remote control

For remote control, the suffix may differ from the number of the corresponding selection used in manual operation. SCPI prescribes that suffix counting starts with 1. Suffix 1 is the default state and used when no specific suffix is specified.

Some standards define a fixed numbering, starting with 0. If the numbering differs in manual operation and remote control, it is indicated for the corresponding command.

Optional mnemonics

Some command systems permit certain mnemonics to be inserted into the header or omitted. These mnemonics are marked by square brackets in the description. The instrument must recognize the long command to comply with the SCPI standard. Some commands are considerably shortened by these optional mnemonics.

Example:

Definition: `HCOPy[:IMMediate]`

Command: `HCOP:IMM` is equivalent to `HCOP`

Optional mnemonics with numeric suffixes

Do not omit an optional mnemonic if it includes a numeric suffix that is relevant for the effect of the command.

Example:

Definition:`DISPlay[:WINDow<1...4>]:MAXimize <Boolean>`

Command: `DISP:MAX ON` refers to window 1.

In order to refer to a window other than 1, you must include the optional `WINDow` parameter with the suffix for the required window.

`DISP:WIND2:MAX ON` refers to window 2.

Parameters

Parameters must be separated from the header by a "white space". If several parameters are specified in a command, they are separated by a comma (,). For a description of the parameter types, refer to chapter 9.1.5.3, "SCPI Parameters", on page 391.

Example:

Definition:`HCOPy:DEVice:CMAP:COLor:RGB <red>,<green>,<blue>`

Command:`HCOP:DEV:CMAP:COL:RGB 3,32,44`

Special characters

<table>
<tr><td>|</td><td>Parameters
A vertical stroke in parameter definitions indicates alternative possibilities in the sense of "or". The effect of the command differs, depending on which parameter is used.
Example:
Definition:HCOPy:PAGE:ORIentation LANDscape | PORTrait
Command HCOP:PAGE:ORI LAND specifies landscape orientation
Command HCOP:PAGE:ORI PORT specifies portrait orientation
Mnemonics
A selection of mnemonics with an identical effect exists for several commands. These mnemonics are indicated in the same line; they are separated by a vertical stroke. Only one of these mnemonics needs to be included in the header of the command. The effect of the command is independent of which of the mnemonics is used.
Example:
DefinitionSENSE:BANDwidth|BWIDth[:RESolution] <numeric_value>
The two following commands with identical meaning can be created:
SENS:BAND:RES 1
SENS:BWID:RES 1</td></tr>
<tr><td>[ ]</td><td>Mnemonics in square brackets are optional and may be inserted into the header or omitted.
Example: HCOPy[:IMMediate]
HCOP:IMM is equivalent to HCOP</td></tr>
<tr><td>{ }</td><td>Parameters in curly brackets are optional and can be inserted once or several times, or omitted.
Example: SENSe:LIST:FREQuency <numeric_value>{,<numeric_value>}
The following are valid commands:
SENS:LIST:FREQ 10
SENS:LIST:FREQ 10,20
SENS:LIST:FREQ 10,20,30,40</td></tr>
</table>

9.1.5.3 SCPI Parameters

Many commands are supplemented by a parameter or a list of parameters. The parameters must be separated from the header by a "white space" (ASCII code 0 to 9, 11 to 32 decimal, e.g. blank). Allowed parameters are:

- Numeric values
- Special numeric values
- Boolean parameters
- Text
- Character strings
- Block data

The parameters required for each command and the allowed range of values are specified in the command description.

Numeric values

Numeric values can be entered in any form, i.e. with sign, decimal point and exponent. Values exceeding the resolution of the instrument are rounded up or down. The mantissa may comprise up to 255 characters, the exponent must lie inside the value range -32000 to 32000. The exponent is introduced by an "E" or "e". Entry of the exponent alone is not allowed. In the case of physical quantities, the unit can be entered. Allowed unit prefixes are G (giga), MA (mega), MOHM and MHZ are also allowed), K (kilo), M (milli), U (micro) and N (nano). If the unit is missing, the basic unit is used.

Example: `SENS:FREQ:STOP 1.5GHz` = `SENS:FREQ:STOP 1.5E9`

Units

For physical quantities, the unit can be entered. Allowed unit prefixes are:

- G (giga)
- MA (mega), MOHM, MHZ
- K (kilo)
- M (milli)
- U (micro)
- N (nano)

If the unit is missing, the basic unit is used.

Example:

`SENSe:FREQ:STOP 1.5GHz` = `SENSe:FREQ:STOP 1.5E9`

Some settings allow relative values to be stated in percent. According to SCPI, this unit is represented by the `PCT` string.

Example:

`HCOP:PAGE:SCAL 90PCT`

Special numeric values

The texts listed below are interpreted as special numeric values. In the case of a query, the numeric value is provided.

- **MIN/MAX**
  MINimum and MAXimum denote the minimum and maximum value.
- **DEF**
  DEFault denotes a preset value which has been stored in the EPROM. This value conforms to the default setting, as it is called by the *RST command.
- **UP/DOWN**
  UP, DOWN increases or reduces the numeric value by one step. The step width can be specified via an allocated step command for each parameter which can be set via UP, DOWN.
- **INF/NINF**

INFinity, Negative INFinity (NINF) represent the numeric values 9.9E37 or -9.9E37, respectively. INF and NINF are only sent as instrument responses.

- **NAN**
  Not A Number (NAN) represents the value 9.91E37. NAN is only sent as a instrument response. This value is not defined. Possible causes are the division of zero by zero, the subtraction of infinite from infinite and the representation of missing values.

Example:

Setting command: `SENSe:LIST:FREQ MAXimum`

Query: `SENS:LIST:FREQ?, Response: 3.5E9`

Queries for special numeric values

The numeric values associated to `MAXimum/MINimum/DEFault` can be queried by adding the corresponding mnemonics to the command. They must be entered following the quotation mark.

Example: `SENSe:LIST:FREQ? MAXimum`

Returns the maximum numeric value as a result.

Boolean Parameters

Boolean parameters represent two states. The "ON" state (logically true) is represented by "ON" or a numeric value 1. The "OFF" state (logically untrue) is represented by "OFF" or the numeric value 0. The numeric values are provided as the response for a query.

Example:

Setting command: `HCOPy:DEV:COL ON`

Query: `HCOPy:DEV:COL?`

Response: `1`

Text parameters

Text parameters observe the syntactic rules for mnemonics, i.e. they can be entered using a short or long form. Like any parameter, they have to be separated from the header by a white space. In the case of a query, the short form of the text is provided.

Example:

Setting command: `HCOPy:PAGE:ORIentation LANDscape`

Query: `HCOP:PAGE:ORI?`

Response: `LAND`

Character strings

Strings must always be entered in quotation marks (' or ").

Example:

`HCOP:ITEM:LABel "Test1"` or `HCOP:ITEM:LABel 'Test1'`

Block data

Block data is a format which is suitable for the transmission of large amounts of data. A command using a block data parameter has the following structure:

Example:

`FORMat:READings:DATA #45168xxxxxxxx`

The ASCII character # introduces the data block. The next number indicates how many of the following digits describe the length of the data block. In the example the 4 following digits indicate the length to be 5168 bytes. The data bytes follow. During the transmission of these data bytes all end or other control signs are ignored until all bytes are transmitted.

#0 specifies a data block of indefinite length. The use of the indefinite format requires a `NL^END` message to terminate the data block. This format is useful when the length of the transmission is not known or if speed or other considerations prevent segmentation of the data into blocks of definite length.

9.1.5.4 Overview of Syntax Elements

The following table provides an overview of the syntax elements:

:	The colon separates the mnemonics of a command. In a command line the separating semicolon marks the uppermost command level.
;	The semicolon separates two commands of a command line. It does not alter the path.
,	The comma separates several parameters of a command.
?	The question mark forms a query.
*****	The asterisk marks a common command.
' ' **"**	Quotation marks introduce a string and terminate it (both single and double quotation marks are possible).
#	The hash symbol introduces binary, octal, hexadecimal and block data. ● Binary: #B10110 ● Octal: #O7612 ● Hexa: #HF3A7 ● Block: #21312
	A "white space" (ASCII-Code 0 to 9, 11 to 32 decimal, e.g. blank) separates the header from the parameters.

9.1.5.5 Structure of a command line

A command line may consist of one or several commands. It is terminated by one of the following:

- a <New Line>

- a <New Line> with EOI
- an EOI together with the last data byte

Several commands in a command line must be separated by a semicolon ";". If the next command belongs to a different command system, the semicolon is followed by a colon.

Example:

`MMEM:COPY "Test1","MeasurementXY";:HCOP:ITEM ALL`

This command line contains two commands. The first command belongs to the MMEM system, the second command belongs to the HCOP system.

If the successive commands belong to the same system, having one or several levels in common, the command line can be abbreviated. To this end, the second command after the semicolon starts with the level that lies below the common levels. The colon following the semicolon must be omitted in this case.

Example:

`HCOP:ITEM ALL;:HCOP:IMM`

This command line contains two commands. Both commands are part of the `HCOP` command system, i.e. they have one level in common.

When abbreviating the command line, the second command begins with the level below HCOP. The colon after the semicolon is omitted. The abbreviated form of the command line reads as follows:

`HCOP:ITEM ALL;IMM`

A new command line always begins with the complete path.

Example:

`HCOP:ITEM ALL`

`HCOP:IMM`

9.1.5.6 Responses to Queries

A query is defined for each setting command unless explicitly specified otherwise. It is formed by adding a question mark to the associated setting command. According to SCPI, the responses to queries are partly subject to stricter rules than in standard IEEE 488.2.

- The requested parameter is transmitted without a header.
  Example: `HCOP:PAGE:ORI?`, Response: `LAND`
- Maximum values, minimum values and all other quantities that are requested via a special text parameter are returned as numeric values.
  Example: `SENSe:FREQuency:STOP? MAX`, Response: `3.5E9`
- Numeric values are output without a unit. Physical quantities are referred to the basic units or to the units set using the `Unit` command. The response `3.5E9` in the previous example stands for 3.5 GHz.
- Truth values (Boolean values) are returned as `0` (for OFF) and `1` (for ON).

Example:
Setting command: `HCOPy:DEV:COL ON`
Query: `HCOPy:DEV:COL?`
Response: `1`

- Text (character data) is returned in a short form.
  Example:
  Setting command: `HCOPy:PAGE:ORIentation LANDscape`
  Query: `HCOP:PAGE:ORI?`
  Response: `LAND`

9.1.6 Command Sequence and Synchronization

IEEE 488.2 defines a distinction between overlapped and sequential commands:

- A sequential command is one which finishes executing before the next command starts executing. Commands that are processed quickly are usually implemented as sequential commands.
- An overlapping command is one which does not automatically finish executing before the next command starts executing. Usually, overlapping commands take longer to process and allow the program to do other tasks while being executed. If overlapping commands do have to be executed in a defined order, e.g. in order to avoid wrong measurement results, they must be serviced sequentially. This is called synchronization between the controller and the instrument.

Setting commands within one command line, even though they may be implemented as sequential commands, are not necessarily serviced in the order in which they have been received. In order to make sure that commands are actually carried out in a certain order, each command must be sent in a separate command line.

Example: Commands and queries in one message

The response to a query combined in a program message with commands that affect the queried value is not predictable.

The following commands always return the specified result:

`:FREQ:STAR 1GHZ;SPAN 100;:FREQ:STAR?`

Result:

`1000000000` (1 GHz)

Whereas the result for the following commands is not specified by SCPI:

`:FREQ:STAR 1GHz;STAR?;SPAN 1000000`

The result could be the value of `STARt` before the command was sent since the instrument might defer executing the individual commands until a program message terminator is received. The result could also be 1 GHz if the instrument executes commands as they are received.

As a general rule, send commands and queries in different program messages.

Example: Overlapping command with *OPC

The instrument implements `INITiate[:IMMediate]` as an overlapped command. Assuming that `INITiate[:IMMediate]` takes longer to execute than `*OPC`, sending the following command sequence results in initiating a sweep and, after some time, setting the `OPC` bit in the `ESR`:

`INIT; *OPC.`

Sending the following commands still initiates a sweep:

`INIT; *OPC; *CLS`

However, since the operation is still pending when the instrument executes `*CLS`, forcing it into the "Operation Complete Command Idle" State (OCIS), `*OPC` is effectively skipped. The `OPC` bit is not set until the instrument executes another `*OPC` command.

9.1.6.1 Preventing Overlapping Execution

To prevent an overlapping execution of commands, one of the commands *OPC, *OPC? or *WAI can be used. All three commands cause a certain action only to be carried out after the hardware has been set. By suitable programming, the controller can be forced to wait for the corresponding action to occur.

*Table 9-2: Synchronization using *OPC, *OPC? and *WAI*

Command	Action	Programming the controller
*OPC	Sets the Operation Complete bit in the ESR after all previous commands have been executed.	• Setting bit 0 in the ESE • Setting bit 5 in the SRE • Waiting for service request (SRQ)
*OPC?	Stops command processing until 1 is returned. This is only the case after the Operation Complete bit has been set in the ESR. This bit indicates that the previous setting has been completed.	Sending *OPC? directly after the command whose processing should be terminated before other commands can be executed.
*WAI	Stops further command processing until all commands sent before *WAI have been executed.	Sending *WAI directly after the command whose processing should be terminated before other commands are executed.

Command synchronization using *WAI or *OPC? appended to an overlapped command is a good choice if the overlapped command takes only little time to process. The two synchronization techniques simply block overlapped execution of the command.

For time consuming overlapped commands it is usually desirable to allow the controller or the instrument to do other useful work while waiting for command execution. Use one of the following methods:

***OPC with a service request**

1. Set the OPC mask bit (bit no. 0) in the ESE: *ESE 1
2. Set bit no. 5 in the SRE: *SRE 32 to enable ESB service request.
3. Send the overlapped command with *OPC

4. Wait for a service request

   The service request indicates that the overlapped command has finished.

***OPC? with a service request**

1. Set bit no. 4 in the SRE: *SRE 16 to enable MAV service request.
2. Send the overlapped command with *OPC?
3. Wait for a service request

   The service request indicates that the overlapped command has finished.

Event Status Register (ESE)

1. Set the OPC mask bit (bit no. 0) in the ESE: *ESE 1
2. Send the overlapped command without *OPC, *OPC? or *WAI
3. Poll the operation complete state periodically (by means of a timer) using the sequence: *OPC; *ESR?

   A return value (LSB) of 1 indicates that the overlapped command has finished.

***OPC? with short timeout**

1. Send the overlapped command without *OPC, *OPC? or *WAI
2. Poll the operation complete state periodically (by means of a timer) using the sequence: <short timeout>; *OPC?
3. A return value (LSB) of 1 indicates that the overlapped command has finished. In case of a timeout, the operation is ongoing.
4. Reset timeout to former value
5. Clear the error queue with `SYStem:ERRor?` to remove the "-410, Query interrupted" entries.

Using several threads in the controller application

As an alternative, provided the programming environment of the controller application supports threads, separate threads can be used for the application GUI and for controlling the instrument(s) via SCPI.

A thread waiting for a *OPC? thus will not block the GUI or the communication with other instruments.

9.1.7 Status Reporting System

The status reporting system stores all information on the current operating state of the instrument, and on errors which have occurred. This information is stored in the status registers and in the error queue. Both can be queried via GPIB bus or LAN interface (`STATus`... commands, see chapter 10.9, "Using the Status Register", on page 721).

9.1.7.1 Hierarchy of status registers

As shown in the following figure, the status information is of hierarchical structure.

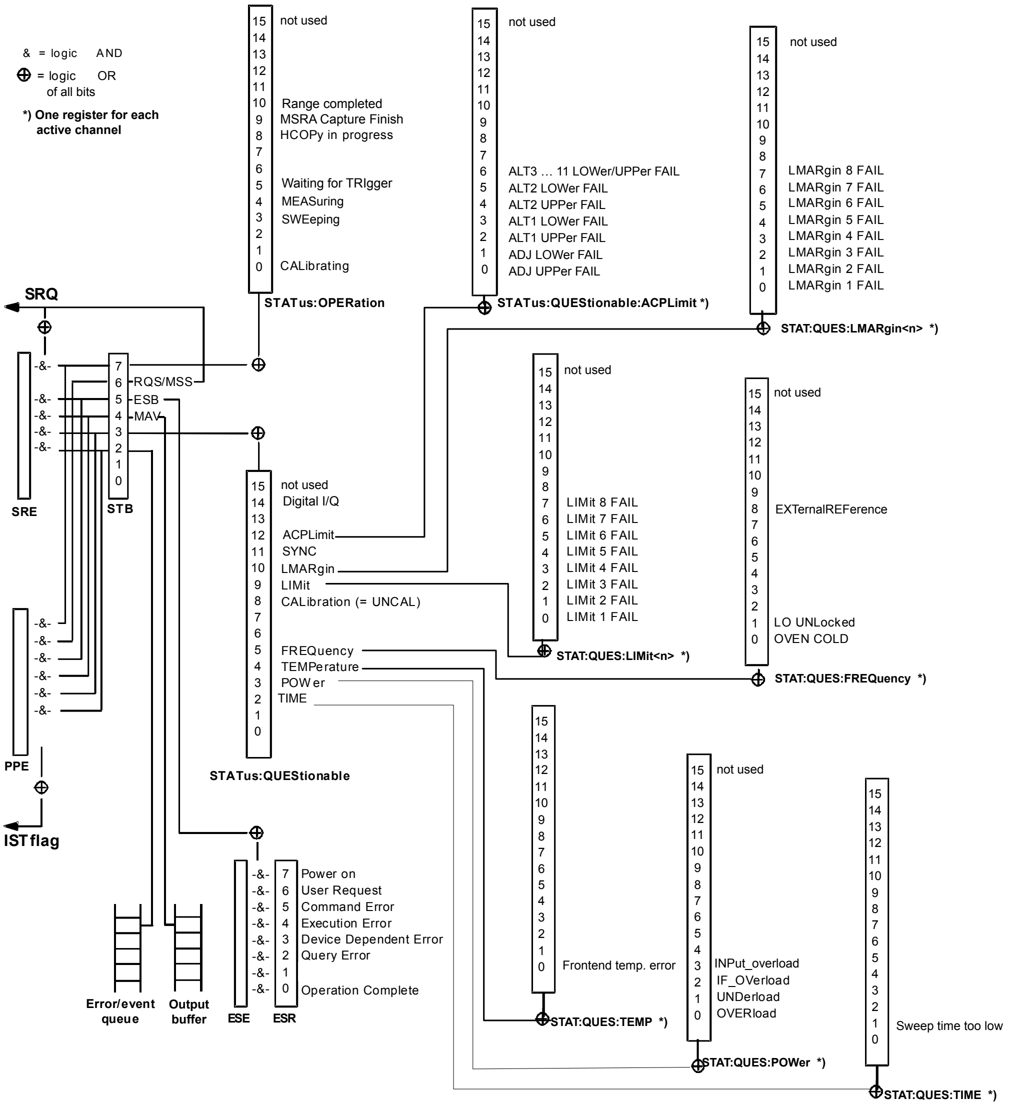


Fig. 9-1: Graphical overview of the R&S FSW status registers hierarchy

- **STB, SRE**
  The STatus Byte (`STB`) register and its associated mask register Service Request Enable (`SRE`) form the highest level of the status reporting system. The `STB` provides a rough overview of the instrument status, collecting the information of the lower-level registers.
- **ESR, SCPI registers**
  The STB receives its information from the following registers:
  – The Event Status Register (`ESR`) with the associated mask register standard Event Status Enable (`ESE`).
  – The `STATus:OPERation` and `STATus:QUEStionable` registers which are defined by SCPI and contain detailed information on the instrument.

- **IST, PPE**
  The `IST` flag ("Individual STatus"), like the `SRQ`, combines the entire instrument status in a single bit. The `PPE` fulfills the same function for the `IST` flag as the `SRE` for the service request.
- **Output buffer**
  The output buffer contains the messages the instrument returns to the controller. It is not part of the status reporting system but determines the value of the `MAV` bit in the `STB` and thus is represented in the overview.

All status registers have the same internal structure.

SRE, ESE

The service request enable register `SRE` can be used as `ENABle` part of the `STB` if the `STB` is structured according to SCPI. By analogy, the `ESE` can be used as the `ENABle` part of the `ESR`.

9.1.7.2 Structure of a SCPI Status Register

Each standard SCPI register consists of 5 parts. Each part has a width of 16 bits and has different functions. The individual bits are independent of each other, i.e. each hardware status is assigned a bit number which is valid for all five parts. Bit 15 (the most significant bit) is set to zero for all parts. Thus the contents of the register parts can be processed by the controller as positive integers.

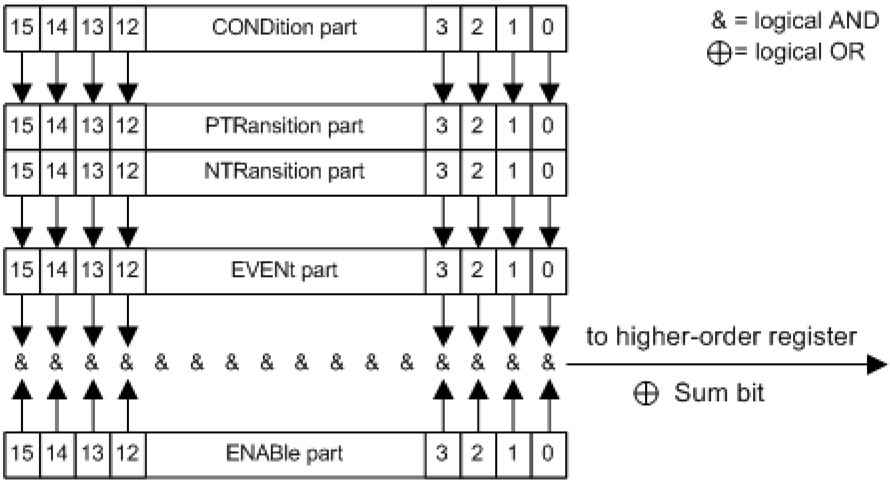


Fig. 9-2: The status-register model

Description of the five status register parts

The five parts of a SCPI register have different properties and functions:

- **CONDition**
  The `CONDition` part is written into directly by the hardware or the sum bit of the next lower register. Its contents reflect the current instrument status. This register part can only be read, but not written into or cleared. Its contents are not affected by reading.

- **PTRansition**
  The two transition register parts define which state transition of the `CONDition` part (none, 0 to 1, 1 to 0 or both) is stored in the `EVENt` part.
  The `Positive-TRansition` part acts as a transition filter. When a bit of the `CONDition` part is changed from 0 to 1, the associated `PTR` bit decides whether the `EVENt` bit is set to 1.
  – PTR bit =1: the `EVENt` bit is set.
  – PTR bit =0: the `EVENt` bit is not set.

  This part can be written into and read as required. Its contents are not affected by reading.
- **NTRansition**
  The `Negative-TRansition` part also acts as a transition filter. When a bit of the `CONDition` part is changed from 1 to 0, the associated `NTR` bit decides whether the `EVENt` bit is set to 1.
  – NTR bit =1: the `EVENt` bit is set.
  – NTR bit =0: the `EVENt` bit is not set.

  This part can be written into and read as required. Its contents are not affected by reading.
- **EVENt**
  The `EVENt` part indicates whether an event has occurred since the last reading, it is the "memory" of the condition part. It only indicates events passed on by the transition filters. It is permanently updated by the instrument. This part can only be read by the user. Reading the register clears it. This part is often equated with the entire register.
- **ENABle**
  The `ENABle` part determines whether the associated `EVENt` bit contributes to the sum bit (see below). Each bit of the `EVENt` part is "ANDed" with the associated `ENABle` bit (symbol '&'). The results of all logical operations of this part are passed on to the sum bit via an "OR" function (symbol '+').
  `ENABle` bit = 0: the associated `EVENt` bit does not contribute to the sum bit
  `ENABle` bit = 1: if the associated `EVENt` bit is "1", the sum bit is set to "1" as well.
  This part can be written into and read by the user as required. Its contents are not affected by reading.

Sum bit

The sum bit is obtained from the `EVENt` and `ENABle` part for each register. The result is then entered into a bit of the `CONDition` part of the higher-order register.

The instrument automatically generates the sum bit for each register. Thus an event can lead to a service request throughout all levels of the hierarchy.

9.1.7.3 Contents of the Status Registers

In the following sections, the contents of the status registers are described in more detail.

Status Byte (STB) and Service Request Enable Register (SRE)

The `STatus Byte` (STB) is already defined in IEEE 488.2. It provides a rough overview of the instrument status by collecting the pieces of information of the lower registers. A special feature is that bit 6 acts as the sum bit of the remaining bits of the status byte.

The STB can thus be compared with the `CONDition` part of an SCPI register and assumes the highest level within the SCPI hierarchy.

The STB is read using the command `*STB?` or a serial poll.

The `STatus Byte` (STB) is linked to the `Service Request Enable` (SRE) register. Each bit of the STB is assigned a bit in the SRE. Bit 6 of the SRE is ignored. If a bit is set in the SRE and the associated bit in the STB changes from 0 to 1, a service request (SRQ) is generated. The SRE can be set using the command `*SRE` and read using the command `*SRE?`.

Table 9-3: Meaning of the bits used in the status byte

Bit No.	Meaning
0...1	Not used
2	Error Queue not empty The bit is set when an entry is made in the error queue. If this bit is enabled by the SRE, each entry of the error queue generates a service request. Thus an error can be recognized and specified in greater detail by polling the error queue. The poll provides an informative error message. This procedure is to be recommended since it considerably reduces the problems involved with remote control.
3	QUEStionable status sum bit The bit is set if an `EVENt` bit is set in the `QUEStionable` status register and the associated `ENABle` bit is set to 1. A set bit indicates a questionable instrument status, which can be specified in greater detail by polling the `QUEStionable` status register.
4	MAV bit (message available) The bit is set if a message is available in the output buffer which can be read. This bit can be used to enable data to be automatically read from the instrument to the controller.
5	ESB bit Sum bit of the event status register. It is set if one of the bits in the event status register is set and enabled in the event status enable register. Setting of this bit indicates a serious error which can be specified in greater detail by polling the event status register.
6	MSS bit (master status summary bit) The bit is set if the instrument triggers a service request. This is the case if one of the other bits of this registers is set together with its mask bit in the service request enable register SRE.
7	`OPERation` status register sum bit The bit is set if an `EVENt` bit is set in the `OPERation` status register and the associated `ENABle` bit is set to 1. A set bit indicates that the instrument is just performing an action. The type of action can be determined by polling the `OPERation` status register.

IST Flag and Parallel Poll Enable Register (PPE)

As with the SRQ, the IST flag combines the entire status information in a single bit. It can be read by means of a parallel poll (see "Parallel Poll" on page 411) or using the command `*IST?`.

The parallel poll enable register (PPE) determines which bits of the STB contribute to the IST flag. The bits of the STB are "ANDed" with the corresponding bits of the PPE, with bit 6 being used as well in contrast to the SRE. The IST flag results from the "ORing" of all results. The PPE can be set using commands `*PRE` and read using command *PRE?.

Event Status Register (ESR) and Event Status Enable Register (ESE)

The ESR is defined in IEEE 488.2. It can be compared with the `EVENt` part of a SCPI register. The event status register can be read out using command *ESR?.

The ESE corresponds to the `ENABle` part of a SCPI register. If a bit is set in the ESE and the associated bit in the ESR changes from 0 to 1, the ESB bit in the STB is set. The ESE register can be set using the command `*ESE` and read using the command `*ESE?`.

Table 9-4: Meaning of the bits used in the event status register

Bit No.	Meaning
0	Operation Complete This bit is set on receipt of the command *OPC exactly when all previous commands have been executed.
1	Not used
2	Query Error This bit is set if either the controller wants to read data from the instrument without having sent a query, or if it does not fetch requested data and sends new instructions to the instrument instead. The cause is often a query which is faulty and hence cannot be executed.
3	Device-dependent Error This bit is set if a device-dependent error occurs. An error message with a number between -300 and -399 or a positive error number, which denotes the error in greater detail, is entered into the error queue.
4	Execution Error This bit is set if a received command is syntactically correct but cannot be performed for other reasons. An error message with a number between -200 and -300, which denotes the error in greater detail, is entered into the error queue.
5	Command Error This bit is set if a command is received, which is undefined or syntactically incorrect. An error message with a number between -100 and -200, which denotes the error in greater detail, is entered into the error queue.
6	User Request This bit is set when the instrument is switched over to manual control.
7	Power On (supply voltage on) This bit is set on switching on the instrument.

STATus:OPERation Register

The `STATus:OPERation` register contains information on current activities of the R&S FSW. It also contains information on activities that have been executed since the last read out.

You can read out the register with `STATus:OPERation:CONDition?` or `STATus:OPERation[:EVENt]?`.

Table 9-5: Meaning of the bits used in the STATus:OPERation register

Bit No.	Meaning
0	`CALibrating` This bit is set as long as the instrument is performing a calibration.
1-2	Not used
3	`SWEeping` Sweep is being performed in base unit (applications are not considered); identical to bit 4 In applications, this bit is not used.
4	`MEASuring` Measurement is being performed in base unit (applications are not considered); identical to bit 3 In applications, this bit is not used.
5	`Waiting for TRIgger` Instrument is ready to trigger and waiting for trigger signal
6-7	Not used
8	`HardCOPy` in progress This bit is set while the instrument is printing a hardcopy.
9	For data acquisition in MSRA mode only: `MSRA capture finish` This bit is set if a data acquisition measurement was completed successfully in MSRA operating mode and data is available for evaluation For details on the MSRA operating mode see the R&S FSW MSRA User Manual.
10	`Range completed` This bit is set when a range in the sweep list has been completed if "Stop after Range" has been activated.
11-14	Not used
15	This bit is always 0.

STATus:QUEStionable Register

The STATus:QUEStionable register contains information on instrument states that do not meet the specifications.

The `STAT:QUES:SYNC` register is used by the applications and is thus described in the individual applications' User Manuals.

You can read out the register with `STAT:QUES:COND` or `STAT:QUES:EVEN`.

The `STATus:QUEStionable` register "sums up" the information from all subregisters (e.g. bit 2 sums up the information for all `STATus:QUEStionable:TIMe` registers). For some subregisters, there may be separate registers for each active channel. Thus, if a status bit in the `STATus:QUEStionable` register indicates an error, the error may have occurred in any of the channel-specific subregisters. In this case, you must check the subregister of each channel to determine which channel caused the error. By default, querying the status of a subregister always returns the result for the currently selected channel.

Table 9-6: Meaning of the bits used in the STATus:QUEStionable register

Bit No.	Meaning
0 - 1	Unused
2	TIMe This bit is set if a time error occurs in any of the active channels. The STATus:QUEStionable:TIMe Register provides more information on the error type.
3	POWer This bit is set if the measured power level in any of the active channels is questionable. The STATus:QUEStionable:POWer Register provides more information on the error type.
4	TEMPerature This bit is set if the temperature is questionable.
5	FREQuency This bit is set if there is anything wrong with the frequency of the local oscillator or the reference frequency in any of the active channels. The STATus:QUEStionable:FREQuency Register provides more information on the error type.
6 - 7	Unused
8	CALibration This bit is set if the R&S FSW is unaligned ("UNCAL" display)
9	LIMit (device-specific) This bit is set if a limit value is violated in any of the active channels in any window. The STATus:QUEStionable:LIMit Register provides more information on the error type.
10	LMARgin (device-specific) This bit is set if a margin is violated in any of the active channels in any window. The STATus:QUEStionable:LMARgin Register provides more information on the error type.
11	SYNC (device-specific) This bit is set if the R&S FSW is not synchronized to the signal that is applied. The R&S FSW is not synchronized if: ● it cannot synchronize to midamble during a measurement or premeasurement ● it cannot find a burst during a measurement or premeasurement ● the results deviate too much from the expected value during premeasurements
12	ACPLimit (device-specific) This bit is set if a limit during ACLR measurements is violated in any of the active channels. The STATus:QUEStionable:ACPLimit Register provides more information on the error type.

Bit No.	Meaning
13	Unused
14	Digital I/Q (device-specific) This bit is set if a connection error occurs at the Digital Baseband Interface (R&S FSW-B17) For details see the R&S FSW I/Q Analyzer User Manual.
15	This bit is always 0.

STATus:QUEStionable:ACPLimit Register

The STATus:QUEStionable:ACPLimit register contains information about the results of a limit check during ACLR measurements. A separate ACPLimit register exists for each active channel.

You can read out the register with`STATus:QUEStionable:ACPLimit:CONDition?` or `STATus:QUEStionable:ACPLimit[:EVENt]?`

Table 9-7: Meaning of the bits used in the STATus:QUEStionable:ACPLimit register

Bit No.	Meaning
0	ADJ UPPer FAIL This bit is set if the limit is exceeded in the **upper adjacent** channel
1	ADJ LOWer FAIL This bit is set if the limit is exceeded in the **lower adjacent** channel.
2	ALT1 UPPer FAIL This bit is set if the limit is exceeded in the **upper 1st alternate** channel.
3	ALT1 LOWer FAIL This bit is set if the limit is exceeded in the **lower 1st alternate** channel.
4	ALT2 UPPer FAIL This bit is set if the limit is exceeded in the **upper 2nd alternate** channel.
5	ALT2 LOWer FAIL This bit is set if the limit is exceeded in the **lower 2nd alternate** channel.
6	ALT3 … 11 LOWer/UPPer FAIL This bit is set if the limit is exceeded in one of the lower or upper alternate channels 3 … 11.
7 to 14	Unused
15	This bit is always 0.

STATus:QUEStionable:FREQuency Register

The STATus:QUEStionable:FREQuency register contains information about the condition of the local oscillator and the reference frequency. A separate frequency register exists for each active channel.

You can read out the register with `STATus:QUEStionable:FREQuency:CONDition?` or `STATus:QUEStionable:FREQuency[:EVENt]?`.

Table 9-8: Meaning of the bits used in the STATus:QUEStionable:FREQuency register

Bit No.	Meaning
0	OVEN COLD This bit is set if the reference oscillator has not yet attained its operating temperature. "OCXO" is displayed.
1	LO UNLocked This bit is set if the local oscillator no longer locks. "LOUNL" is displayed.
2 to 7	Not used
8	EXTernalREFerence This bit is set if you have selected an external reference oscillator but did not connect a useable external reference source. In that case the synthesizer can not lock. The frequency in all probability is not accurate.
9 to 14	Not used
15	This bit is always 0.

STATus:QUEStionable:LIMit Register

The STATus:QUEStionable:LIMit register contains information about the results of a limit check when you are working with limit lines.

A separate LIMit register exists for each active channel and for each window.

You can read out the register with `STATus:QUEStionable:LIMit<n>:CONDition?` or `STATus:QUEStionable:LIMit<n>[:EVENt]?`.

Table 9-9: Meaning of the bits used in the STATus:QUEStionable:LIMit register

Bit No.	Meaning
0	LIMit 1 FAIL This bit is set if limit line 1 is violated.
1	LIMit 2 FAIL This bit is set if limit line 2 is violated.
2	LIMit 3 FAIL This bit is set if limit line 3 is violated.
3	LIMit 4 FAIL This bit is set if limit line 4 is violated.
4	LIMit 5 FAIL This bit is set if limit line 5 is violated.
5	LIMit 6 FAIL This bit is set if limit line 6 is violated.
6	LIMit 7 FAIL This bit is set if limit line 7 is violated.
7	LIMit 8 FAIL This bit is set if limit line 8 is violated.

Bit No.	Meaning
8 to 14	Unused
15	This bit is always 0.

STATus:QUEStionable:LMARgin Register

This register contains information about the observance of limit margins.

A separate LMARgin register exists for each active channel and for each window.

It can be read using the commands
`STATus:QUEStionable:LMARgin:CONDition?` and
`STATus:QUEStionable:LMARgin[:EVENt]?`.

Table 9-10: Meaning of the bits used in the STATus:QUEStionable:LMARgin register

Bit No.	Meaning
0	LMARgin 1 FAIL This bit is set if limit margin 1 is violated.
1	LMARgin 2 FAIL This bit is set if limit margin 2 is violated.
2	LMARgin 3 FAIL This bit is set if limit margin 3 is violated.
3	LMARgin 4 FAIL This bit is set if limit margin 4 is violated.
4	LMARgin 5 FAIL This bit is set if limit margin 5 is violated.
5	LMARgin 6 FAIL This bit is set if limit margin 6 is violated.
6	LMARgin 7 FAIL This bit is set if limit margin 7 is violated.
7	LMARgin 8 FAIL This bit is set if limit margin 8 is violated.
8 to 14	Not used
15	This bit is always 0.

STATus:QUEStionable:POWer Register

The `STATus:QUEStionable:POWer` register contains information about possible overload situations that may occur during operation of the R&S FSW. A separate power register exists for each active channel.

You can read out the register with `STATus:QUEStionable:POWer:CONDition?` or `STATus:QUEStionable:POWer[:EVENt]?`

Table 9-11: Meaning of the bits used in the STATus:QUEStionable:POWer register

Bit No.	Meaning
0	OVERload This bit is set if an overload occurs at the RF input, causing signal distortion but not yet causing damage to the device. The R&S FSW displays the keyword "OVLD".
1	UNDerload This bit is set if an underload occurs at the RF input. The R&S FSW displays the keyword "UNLD".
2	IF_OVerload This bit is set if an overload occurs in the IF path. The R&S FSW displays the keyword "IFOVL".
3	Input Overload This bit is set if the signal level at the RF input connector exceeds the maximum (see chapter 5.2.1.1, "RF Input Protection", on page 161). The RF input is disconnected from the input mixer to protect the device. In order to re-enable measurement, decrease the level at the RF input connector and reconnect the RF input to the mixer input (see `INPut:ATTenuation:PROTection:RESet` on page 595). The R&S FSW displays the keyword "INPUT OVLD".
4 to 14	Unused
15	This bit is always 0.

STATus:QUEStionable:TEMPerature Register

The STATus:QUEStionable:TEMPerature register contains information about possible temperature deviations that may occur during operation of the R&S FSW. A separate temperature register exists for each active channel.

Table 9-12: Meaning of the bits used in the STATus:QUEStionable:TEMPerature register

Bit No.	Meaning
0	This bit is set if the frontend temperature sensor deviates by a certain degree from the self-alignment temperature. During warmup, this bit is always 0. For details see "Temperature check" on page 342.
1 to 14	Unused
15	This bit is always 0.

STATus:QUEStionable:TIMe Register

The STATus:QUEStionable:TIMe register contains information about possible time errors that may occur during operation of the R&S FSW. A separate time register exists for each active channel.

Table 9-13: Meaning of the bits used in the STATus:QUEStionable:TIMe register

Bit No.	Meaning
0	not used
1	Sweep time too low This bit is set if the sweep time is too low and thus calibration fails. Note: the STATus:QUEStionable bit for CALibration is not affected by this error.
2 to 14	Unused
15	This bit is always 0.

9.1.7.4 Application of the Status Reporting System

The purpose of the status reporting system is to monitor the status of one or several devices in a measuring system. To do this and react appropriately, the controller must receive and evaluate the information of all devices. The following standard methods are used:

- **Service request** (SRQ) initiated by the instrument
- **Serial poll** of all devices in the bus system, initiated by the controller in order to find out who sent a SRQ and why
- **Parallel poll** of all devices
- Query of a **specific instrument status** by means of commands
- Query of the **error queue**

Service Request

Under certain circumstances, the instrument can send a service request (SRQ) to the controller. Usually this service request initiates an interrupt at the controller, to which the control program can react appropriately. As evident from figure 9-1, an SRQ is always initiated if one or several of bits 2, 3, 4, 5 or 7 of the status byte are set and enabled in the SRE. Each of these bits combines the information of a further register, the error queue or the output buffer. The `ENABle` parts of the status registers can be set such that arbitrary bits in an arbitrary status register initiate an SRQ. In order to make use of the possibilities of the service request effectively, all bits should be set to "1" in enable registers SRE and ESE.

Use of the command *OPC to generate an SRQ at the end of a sweep

1. `CALL InstrWrite(analyzer, "*ESE 1")` 'Set bit 0 in the ESE (Operation Complete)
2. `CALL InstrWrite(analyzer, "*SRE 32")` 'Set bit 5 in the SRE (ESB)
3. `CALL InstrWrite(analyzer, "*INIT;*OPC")` ' Generate an SRQ after operation complete

   After its settings have been completed, the instrument generates an SRQ.

The SRQ is the only possibility for the instrument to become active on its own. Each controller program should cause the instrument to initiate a service request if errors occur. The program should react appropriately to the service request.

A detailed example for a service request routine can be found in chapter 10.12.1, "Service Request", on page 762.

Serial Poll

In a serial poll, just as with command *STB, the status byte of an instrument is queried. However, the query is realized via interface messages and is thus clearly faster.

The serial poll method is defined in IEEE 488.1 and used to be the only standard possibility for different instruments to poll the status byte. The method also works for instruments which do not adhere to SCPI or IEEE 488.2.

The serial poll is mainly used to obtain a fast overview of the state of several instruments connected to the controller.

Parallel Poll

In a parallel poll, up to eight instruments are simultaneously requested by the controller using a single command to transmit 1 bit of information each on the data lines, i.e., to set the data line allocated to each instrument to a logical "0" or "1".

In addition to the SRE register, which determines the conditions under which an SRQ is generated, there is a Parallel Poll Enable register (PPE) which is ANDed with the STB bit by bit, considering bit 6 as well. This register is ANDed with the STB bit by bit, considering bit 6 as well. The results are ORed, the result is possibly inverted and then sent as a response to the parallel poll of the controller. The result can also be queried without parallel poll using the command *IST?.

The instrument first has to be set for the parallel poll using the command PPC. This command allocates a data line to the instrument and determines whether the response is to be inverted. The parallel poll itself is executed using PPE.

The parallel poll method is mainly used to find out quickly which one of the instruments connected to the controller has sent a service request. To this effect, SRE and PPE must be set to the same value.

Query of an instrument status

Each part of any status register can be read using queries. There are two types of commands:

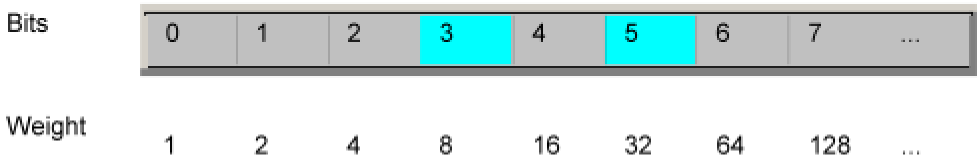
- The common commands *ESR?, *IDN?, *IST?, *STB? query the higher-level registers.
- The commands of the STATus system query the SCPI registers (STATus:QUEStionable...)

The returned value is always a decimal number that represents the bit pattern of the queried register. This number is evaluated by the controller program.

Queries are usually used after an SRQ in order to obtain more detailed information on the cause of the SRQ.

Decimal representation of a bit pattern

The STB and ESR registers contain 8 bits, the SCPI registers 16 bits. The contents of a status register are specified and transferred as a single decimal number. To make this possible, each bit is assigned a weighted value. The decimal number is calculated as the sum of the weighted values of all bits in the register that are set to 1.



Example:

The decimal value 40 = 32 + 8 indicates that bits no. 3 and 5 in the status register (e.g. the `QUEStionable` status summary bit and the `ESB` bit in the `STatus` Byte) are set.

Error Queue

Each error state in the instrument leads to an entry in the error queue. The entries of the error queue are detailed plain text error messages that can be looked up in the Error Log or queried via remote control using `SYSTem:ERRor[:NEXT]?` or `SYSTem:ERRor:ALL?`. Each call of `SYSTem:ERRor[:NEXT]?` provides one entry from the error queue. If no error messages are stored there any more, the instrument responds with 0, "No error".

The error queue should be queried after every SRQ in the controller program as the entries describe the cause of an error more precisely than the status registers. Especially in the test phase of a controller program the error queue should be queried regularly since faulty commands from the controller to the instrument are recorded there as well.

9.1.7.5 Reset Values of the Status Reporting System

The following table contains the different commands and events causing the status reporting system to be reset. None of the commands, except `*RST` and `SYSTem:PRESet`, influence the functional instrument settings. In particular, `DCL` does not change the instrument settings.

Table 9-14: Resetting the status reporting system

Event	**Switching on supply voltage** **Power-On-Status-Clear**		**DCL, SDC** **(Device Clear, Selected Device Clear)**	***RST or SYSTem:PRESet**	**STATus:PRESet**	***CLS**
Effect	**0**	**1**				
Clear STB, ESR	-	yes	-	-	-	yes
Clear SRE, ESE	-	yes	-	-	-	-
Clear PPE	-	yes	-	-	-	-

Event	Switching on supply voltage Power-On-Status-Clear		DCL, SDC (Device Clear, Selected Device Clear)	*RST or SYSTem:PRESet	STATus:PRESet	*CLS
Effect	**0**	**1**				
Clear EVENt parts of the registers	-	yes	-	-	-	yes
Clear ENABle parts of all OPERation and QUEStionable registers; Fill ENABle parts of all other registers with "1".	-	yes	-	-	yes	-
Fill PTRansition parts with "1"; Clear NTRansition parts	-	yes	-	-	yes	-
Clear error queue	yes	yes	-	-	-	yes
Clear output buffer	yes	yes	yes	1)	1)	1)
Clear command processing and input buffer	yes	yes	yes	-	-	-
1) The first command in a command line that immediately follows a <PROGRAM MESSAGE TERMINATOR> clears the output buffer.						

9.1.8 General Programming Recommendations

Initial instrument status before changing settings

Manual operation is designed for maximum possible operating convenience. In contrast, the priority of remote control is the "predictability" of the instrument status. Thus, when a command attempts to define incompatible settings, the command is ignored and the instrument status remains unchanged, i.e. other settings are not automatically adapted. Therefore, control programs should always define an initial instrument status (e.g. using the *RST command) and then implement the required settings.

Command sequence

As a general rule, send commands and queries in different program messages. Otherwise, the result of the query may vary depending on which operation is performed first (see also chapter 9.1.6.1, "Preventing Overlapping Execution", on page 397).

Reacting to malfunctions

The service request is the only possibility for the instrument to become active on its own. Each controller program should instruct the instrument to initiate a service request in case of malfunction. The program should react appropriately to the service request.

Error queues

The error queue should be queried after every service request in the controller program as the entries describe the cause of an error more precisely than the status registers. Especially in the test phase of a controller program the error queue should be queried regularly since faulty commands from the controller to the instrument are recorded there as well.

9.2 GPIB Languages

The R&S FSW analyzer family supports a subset of the GPIB commands used by other devices. Thus it can emulate other devices in order to use existing remote control programs.

The device model to be emulated is selected manually using "SETUP > Network + Remote > GPIB tab > Language". Via the GPIB interface using the `SYSTem:LANGuage` on page 726 command.

In order to emulate device models that are not part of the selection list of the GPIB "Language" setting, you can modify the identification string received in response to the ID command ("Identification String" setting). Thus, any device model whose command set is compatible with one of the supported device models can be emulated.

Supported languages

Language	Comment
SCPI	
71100C	Compatible to 8566A/B
71200C	Compatible to 8566A/B
71209A	Compatible to 8566A/B
8560E	
8561E	
8562E	
8563E	
8564E	
8565E	
8566A	Command sets A and B are available. Command sets A and B differ in the rules regarding the command structure.
8566B	
8568A	Command sets A and B are available. Command sets A and B differ in the rules regarding the command structure.
8568A_DC	Uses DC input coupling by default if supported by the instrument

Language	Comment
8568B	Command sets A and B are available. Command sets A and B differ in the rules regarding the command structure.
8568B_DC	Uses DC input coupling by default if supported by the instrument
8591E	Compatible to 8594E
8594E	Command sets A and B are available. Command sets A and B differ in the rules regarding the command structure.
FSEA	
FSEB	
FSEM	
FSEK	
PSA89600	

Notes:

- If you select a language other than "SCPI", the GPIB address is set to 18 if it was 20 before.
- The Start/stop frequency, reference level and number of sweep points are adapted to the selected instrument model.
- When you switch between remote control languages, the following settings or changes are made:
  SCPI:
  The instrument performs a PRESET.
  8566A/B, 8568A/B, 8594E; FSEA, FSEB, FSEM; FSEK:
  – The instrument performs a PRESET.
  – The following instrument settings are changed:

Table 9-15: Instrument settings for emulation of 8566A/B, 8568A/B, 8594E; FSEA, FSEB, FSEM; FSEK instruments

Model	# of Trace Points	Start Freq.	Stop Freq.	Ref Level	Input Coupling
8566A/B	1001	2 GHz	22 GHz	0 dBm	AC
8568A/B	1001	0 Hz	1.5 GHz	0 dBm	AC
8560E	601	0 Hz	2.9 GHz	0 dBm	AC
8561E	601	0 Hz	6.5 GHz	0 dBm	AC
8562E	601	0 Hz	13.2 GHz	0 dBm	AC
8563E	601	0 Hz	26.5 GHz	0 dBm	AC
8564E	601	0 Hz	40 GHz	0 dBm	AC
8565E	601	0 Hz	50 GHz	0 dBm	AC
8594E	401	0 Hz	3 GHz	0 dBm	AC
FSEA	500	0 Hz	3.5 GHz	-20 dBm	AC

Model	# of Trace Points	Start Freq.	Stop Freq.	Ref Level	Input Coupling
FSEB	500	0 Hz	7 GHz	-20 dBm	AC
FSEM	500	0 Hz	26.5 GHz	-20 dBm	AC
FSEK	500	0 Hz	40 GHz	-20 dBm	AC

Note: The stop frequency indicated in the table may be limited to the corresponding frequency of the R&S FSW, if required.

9.3 The IECWIN Tool

The R&S FSW is delivered with *IECWIN* installed, an auxiliary tool provided free of charge by R&S. IECWIN is a program to send SCPI commands to a measuring instrument either interactively or from a command script.

The R&S IECWIN32 tool is provided free of charge. The functionality may change in a future version without notice.

IECWIN offers the following features:

- Connection to instrument via several interfaces/protocols (GPIB, VISA, named pipe (if IECWIN is run on the instrument itself), RSIB)
- Interactive command entry
- Browsing available commands on the instrument
- Error checking following every command
- Execution of command scripts
- Storing binary data to a file
- Reading binary data from a file
- Generation of a log file

For command scripts, IECWIN offers the following features:

- Synchronization with the instrument on every command
- Checking expected result for query commands (as string or numeric value)
- Checking for expected errors codes
- Optional pause on error
- Nested command scripts
- Single step mode
- Conditional execution, based on the *IDN and *OPT strings

You can use the IECWIN to try out the programming examples provided in the R&S FSW User Manuals.

Starting IECWIN

IECWIN is available from the Windows task bar on the R&S FSW, or by executing the following file:

`C:\Program Files (x86)\Rohde-Schwarz\FSW\<1.10 or higher>\iecwin32.exe`

You can also copy the program to any Windows PC or laptop. Simply copy the `iecwin32.exe`, `iecwin.chm` and `rsib32.dll` files from the location above to the same folder on the target computer.

When the tool is started, a "Connection settings" dialog box is displayed. Define the connection from the computer the IECWIN tool is installed on to the R&S FSW you want to control. If you are using the tool directly on the R&S FSW, you can use an NT Pipe (COM Parser) connection, which requires no further configuration. For help on setting up other connection types, check the tool's online help (by clicking the "Help" button in the dialog box) or refer to chapter 9.1, "Remote Control Basics", on page 382.

The IECWIN offers an online help with extensive information on how to work with the tool.

9.4 Network and Remote Control Settings

The network and remote control settings are defined in the "Network + Remote" dialog box which is displayed when you press the SETUP key and then select "Network + Remote".

The remote commands required to define these settings are described in chapter 10.8.6, "Configuring the Network and Remote Control", on page 715.

Step-by-step instructions are provided in chapter 9.5, "How to Set Up a Network and Remote Control", on page 425.

9.4.1 General Network Settings

The R&S FSW can be operated in a local area network (LAN), for example to control the instrument from a remote PC or use a network printer.

The general network settings are defined in the "Network" tab of the "Network + Remote" dialog box.

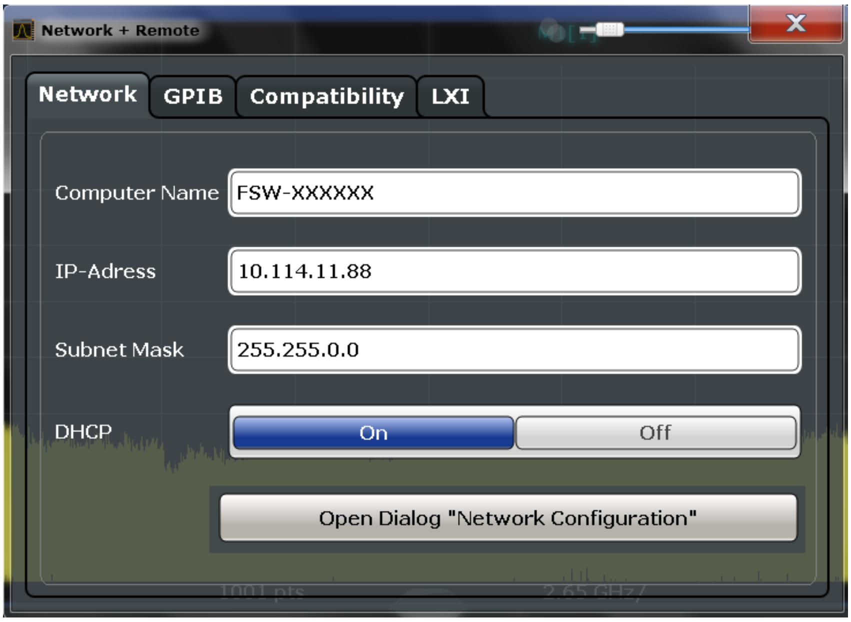
NOTICE

Risk of network problems

All parameters can be edited here; however, beware that changing the computer name has major effects in a network. For details, see chapter 9.5.1, "How to Configure a Network", on page 426.

For step-by-step instructions see chapter 9.5, "How to Set Up a Network and Remote Control", on page 425.

For details on setting up the R&S FSW for remote control, see chapter 9.5, "How to Set Up a Network and Remote Control", on page 425.



Computer Name

Each instrument is delivered with an assigned computer name, but this name can be changed. The naming conventions of Windows apply. If too many characters and/or numbers are entered, an error message is displayed in the status line.

IP Address

Defines the IP address. The TCP/IP protocol is preinstalled with the IP address 10.0.0.10. If the DHCP server is available ("DHCP On"), the setting is read-only.

The IP address consists of four number blocks separated by dots. Each block contains 3 numbers in maximum (e.g. 100.100.100.100), but also one or two numbers are allowed in a block (as an example see the preinstalled address).

Subnet Mask

Defines the subnet mask. The TCP/IP protocol is preinstalled with the subnet mask 255.255.255.0. If the DHCP server is available ("DHCP On"), this setting is read-only.

The subnet mask consists of four number blocks separated by dots. Each block contains 3 numbers in maximum (e.g. 100.100.100.100), but also one or two numbers are allowed in a block (as an example see the preinstalled address).

DHCP

Switches between DHCP server available (On) or not available (Off). If a DHCP server is available in the network, the IP address and subnet mask of the instrument are obtained automatically from the DHCP server.

Network Configuration

Opens the standard Windows "Network Configuration" dialog box for further configuration.

9.4.2 GPIB Settings

Alternatively to connecting the R&S FSW to a LAN, the GPIB interface can be used to connect a remote PC. For details see chapter 9.1.1.2, "GPIB Interface (IEC 625/IEEE 418 Bus Interface)", on page 385).

The GPIB settings are defined in the "GPIB" tab of the "Network + Remote" dialog box.

GPIB Address

Defines the GPIB address. Values from 0 to 30 are allowed. The default address is 20.

SCPI command:

SYSTem:COMMunicate:GPIB[:SELF]:ADDRess on page 715

Identification String

Defines the identification string for the device which is provided as a response to the *IDN? query. Maximum 36 characters are allowed.

SCPI command:

SYSTem:IDENtify[:STRing] on page 716

Reset to Factory String

Restores the default identification string. Each instrument has a unique ID according to the following syntax:

Rohde&Schwarz,FSW,<Unique number>

SCPI command:

SYSTem:IDENtify:FACTory on page 716

Remote Display Update

Defines whether the instrument display is updated or switched off when changing from manual operation to remote control.

SCPI command:

SYSTem:DISPlay:UPDate on page 716

GPIB Terminator

Changes the GPIB receive terminator.

According to the standard, the terminator in ASCII is <LF> and/or <EOI>. For binary data transfers (e.g. trace data) from the control computer to the instrument, the binary code used for <LF> might be included in the binary data block, and therefore should not be interpreted as a terminator in this particular case. This can be avoided by changing the receive terminator to EOI.

SCPI command:

SYSTem:COMMunicate:GPIB[:SELF]:RTERminator on page 715

***IDN Format**

Defines the response format to the remote command *IDN? (see *IDN? on page 446). This function is intended for re-use of existing control programs together with the R&S FSW.

"Leg"	Legacy format, compatible to the R&S FSP/FSU/FSQ family
"New"	R&S FSW format

SCPI command:

SYSTem:FORMat:IDENt on page 720

I/O Logging

Activates or deactivates the SCPI error log function. All remote control commands received by the R&S FSW are recorded in the following log file:

`C:\R_S\instr\ScpiLogging\ScpiLog.txt`

Logging the commands may be extremely useful for debug purposes, e.g. in order to find misspelled keywords in control programs.

SCPI command:

`SYSTem:CLOGging` on page 701

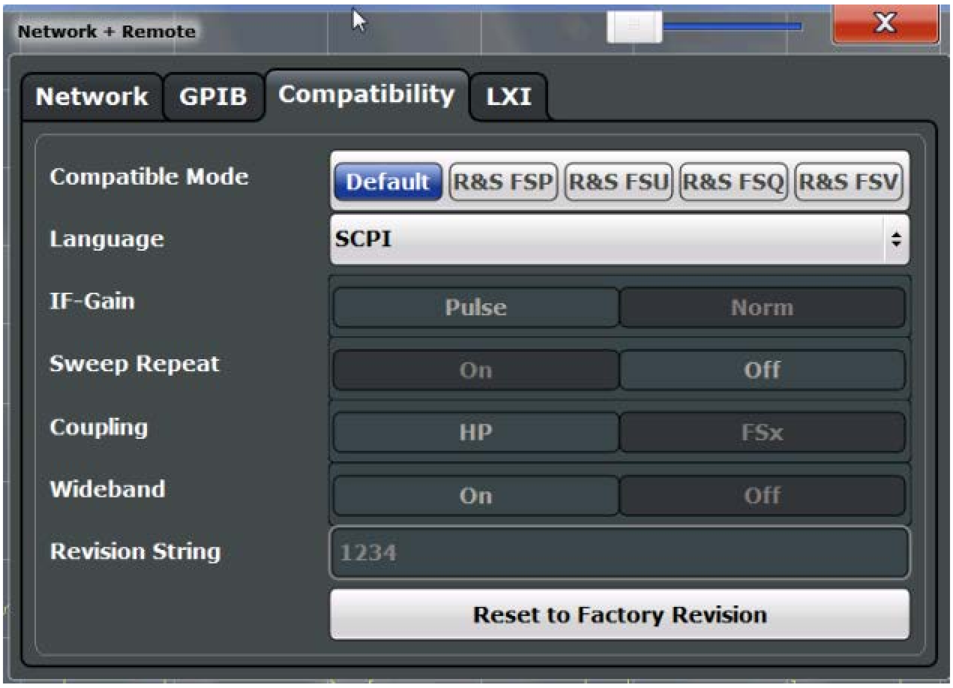
9.4.3 Compatibility Settings

The R&S FSW can emulate the GPIB interface of other signal and spectrum analyzers, e.g. in order to use existing control applications.

Compatibility with former R&S signal and spectrum analyzers

As a rule, the R&S FSW supports most commands from previous R&S signal and spectrum analyzers such as the FSQ, FSP, FSU, or FSV. However, the default values, in particular the number of sweep points or particular bandwidths, may vary. Therefore, the R&S FSW can emulate these other devices, including their default values, in order to repeat previous measurements or support existing control applications as in legacy systems.

The required settings are configured in the "Compatibility" tab of the "Network +Remote" dialog box.



Compatible Mode

Sets the R&S FSW in a state compatible to previous R&S devices. In particular, the number of measurement points and available bandwidths are adjusted to those of other devices.

"Default"	Standard R&S FSW operation; Resets the number of measurement points and available bandwidths to default R&S FSW values.
"R&S FSP"	Sets the number of measurement points and available bandwidths as in R&S FSP devices.
"R&S FSU"	Sets the number of measurement points and available bandwidths as in R&S FSU devices.
"R&S FSQ"	Sets the number of measurement points and available bandwidths as in R&S FSQ devices.
"R&S FSV"	Sets the number of measurement points and available bandwidths as in R&S FSV devices.

SCPI command:
`SYSTem:COMPatible` on page 715

Language

Defines the remote-control language to be used.

For details on the GPIB languages, see chapter 10.10.2, "Reference: GPIB Commands of Emulated HP Models", on page 728.

Note: For PSA89600 emulation, the option is indicated as "B7J" for the `*OPT?` query ("B7J, 140" if Wideband is activated).

SCPI command:
`SYSTem:LANGuage` on page 726

IF Gain

Configures the internal IF gain settings in HP emulation mode due to the application needs. This setting is only taken into account for resolution bandwidth < 300 kHz.

NORM	Optimized for high dynamic range, overload limit is close to reference level.
PULS	Optimized for pulsed signals, overload limit up to 10 dB above reference level.

This setting is only available if an HP language is selected (see "Language" on page 422).

SCPI command:
`SYSTem:IFGain:MODE` on page 726

Sweep Repeat
Controls a repeated sweep of the E1 and MKPK HI HP model commands (for details on the commands refer to chapter 10.10.2, "Reference: GPIB Commands of Emulated HP Models", on page 728). If the repeated sweep is OFF, the marker is set without sweeping before.

Note: In single sweep mode, switch off this setting before you set the marker via the E1 and MKPK HI commands in order to avoid sweeping again.

This setting is only available if a HP language is selected (see "Language" on page 422).

SCPI command:
SYSTem:RSW on page 728

Coupling
Controls the default coupling ratios in the HP emulation mode for:

- span and resolution bandwidth (Span/RBW)
- resolution bandwidth and video bandwidth (RBW/VBW)

For FSx, the standard parameter coupling of the instrument is used. As a result, in most cases a shorter sweeptime is used than in case of HP.

This setting is only available if a HP language is selected (see "Language" on page 422).

SCPI command:
SYSTem:HPCoupling on page 725

Wideband
This setting is only available for PSA89600 emulation.

If activated, the option is indicated as "B7J, 140" for the *OPT? query.

If deactivated, "B7J".

SCPI command:
SYST:PSA:WIDeband on page 727

Revision String
Defines the response to the REV? query for the revision number (HP emulation only, see "GPIB Address" on page 420). Max. 36 characters are allowed.

SCPI command:
SYSTem:REVision[:STRing] on page 727

Resetting the Factory Revision
Resets the response to the REV? query for the revision number to the factory default (HP emulation only, see "Language" on page 422).

SCPI command:
SYSTem:REVision:FACTory on page 727

9.4.4 LXI Settings

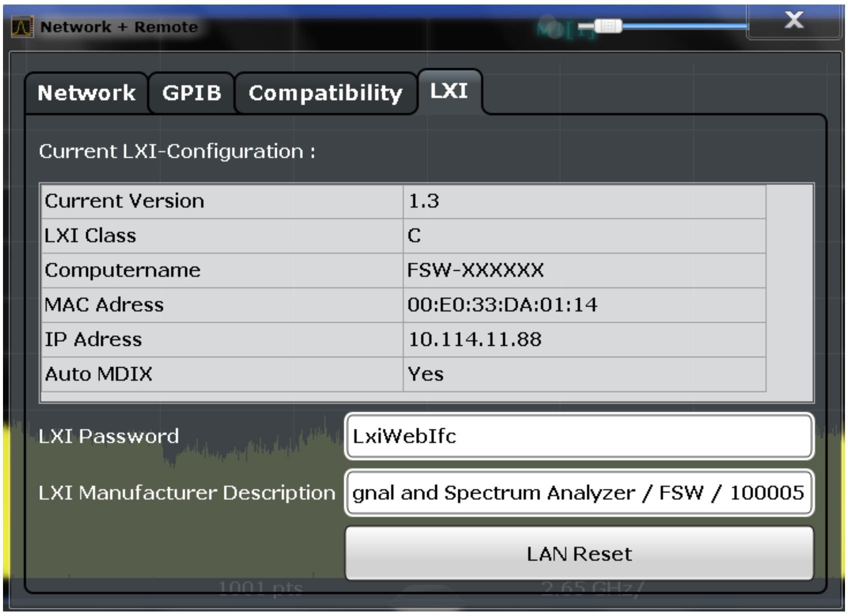
On the R&S FSW the LXI Class C functionality is already installed and enabled; thus, the instrument can be accessed via any web browser (e.g. the Microsoft Internet Explorer) to perform the following tasks:

- modifying network configurations
- modifying device configurations
- monitoring connections from the device to other devices

The "LXI" tab of the "Network + Remote" dialog box provides basic LXI functions and information for the R&S FSW.

Alternatively, you can change the LAN settings using the LXI Web browser interface, see chapter 9.5.1.4, "How to Configure the LAN Using the LXI Browser Interface", on page 429.

Only user accounts with administrator rights are able to use LXI functionality.



Current LXI Configuration

Displays the current LXI information from the R&S FSW (read-only).

"Current version"	Current LXI version
"LXI Class"	LXI device class
"Computer name"	Name of the R&S FSW as defined in the operating system (see also "Computer Name" on page 418)

"MAC address"	Media Access Control address (MAC address), a unique identifier for the network card in the R&S FSW
"IP address"	IP address of the R&S FSW as defined in the operating system (see also "IP Address" on page 418).
"Auto MDIX"	Enables the use of the built-in Auto-MDI(X) Ethernet functionality.

SCPI command:

`SYSTem:LXI:INFo?` on page 716

LXI Password

Password for LAN configuration. The default password is *LxiWebIfc*.

SCPI command:

`SYSTem:LXI:PASSword` on page 717

LXI Manufacturer Description

Instrument description of the R&S FSW

SCPI command:

`SYSTem:LXI:MDEScription` on page 717

LAN Reset

Resets the LAN configuration to its default settings (LCI function).

According to the LXI standard, an LCI must set the following parameters to a default state.

Parameter	Value
TCP/IP Mode	DHCP + Auto IP Address
Dynamic DNS	Enabled
ICMP Ping	Enabled
Password for LAN configuration	LxiWebIfc

The LAN settings are configured in the "Network" tab of the "Network + Remote" dialog box or using the instrument's LXI Browser interface.

SCPI command:

`SYSTem:LXI:LANReset` on page 717

9.5 How to Set Up a Network and Remote Control

NOTICE

Risk of network failure

Before connecting the instrument to the network or configuring the network, consult your network administrator. Errors may affect the entire network.

Remote operation

Remote operation of the instrument from a connected computer is possible using SCPI commands (see chapter 9.1.2, "SCPI (Standard Commands for Programmable Instruments)", on page 386). Sending remote commands requires the instrument to be configured in a LAN network or connected to a PC via the GPIB interface as described in chapter 9.5.1, "How to Configure a Network", on page 426.

Remote Desktop

In production test and measurement, a common requirement is central monitoring of the T&M instruments for remote maintenance and remote diagnostics. Equipped with the Remote Desktop software of Windows, the R&S FSW ideally meets requirements for use in production. The computer that is used for remote operation is called "controller" here. The following tasks can be performed using Remote Desktop:

- Access to the control functions via a virtual front panel (soft front panel)
- Printout of measurement results directly from the controller
- Storage of measured data on the controller's hard disk

This documentation provides basic instructions on setting up the Remote Desktop for the R&S FSW. For details refer to the Windows operating system documentation.

9.5.1 How to Configure a Network

A precondition for operating or monitoring the R&S FSW remotely is that it is connected to a LAN network or a PC connected to the GPIB interface. This is described here.

Windows Firewall Settings

A firewall protects an instrument by preventing unauthorized users from gaining access to it through a network. Rohde & Schwarz highly recommends the use of the firewall on your instrument. R&S instruments are shipped with the Windows firewall enabled and preconfigured in such a way that all ports and connections for remote control are enabled. For more details on firewall configuration see the R&S White Paper "Malware Protection" (available at http://www.rohde-schwarz.com/appnote/1EF73) and the Windows help system.

9.5.1.1 How to Connect the Instrument to the Network

There are two methods to establish a LAN connection to the instrument:

- A non-dedicated network (Ethernet) connection from the instrument to an existing network made with an ordinary RJ-45 network cable. The instrument is assigned an IP address and can coexist with a computer and with other hosts on the same network.
- A dedicated network connection (Point-to-point connection) between the instrument and a single computer made with a (crossover) RJ-45 network cable. The computer must be equipped with a network adapter and is directly connected to the instrument. The use of hubs, switches, or gateways is not required, however, data transfer is still

performed using the TCP/IP protocol. An IP address has to be assigned to the instrument and the computer, see chapter 9.5.1.2, "How to Assign the IP Address", on page 427.
Note: As the R&S FSW uses a 1 GBit LAN, a crossover cable is not necessary (due to Auto-MDI(X) functionality).

► To establish a non-dedicated network connection, connect a commercial RJ-45 cable to one of the LAN ports.
To establish a dedicated connection, connect a (crossover) RJ-45 cable between the instrument and a single PC.

If the instrument is connected to the LAN, Windows automatically detects the network connection and activates the required drivers.

The network card can be operated with a 1 GBit Ethernet IEEE 802.3u interface.

9.5.1.2 How to Assign the IP Address

Depending on the network capacities, the TCP/IP address information for the instrument can be obtained in different ways.

- If the network supports dynamic TCP/IP configuration using the Dynamic Host Configuration Protocol (DHCP), all address information can be assigned automatically.
- If the network does not support DHCP, or if the instrument is set to use alternate TCP/IP configuration, the addresses must be set manually.

By default, the instrument is configured to use dynamic TCP/IP configuration and obtain all address information automatically. This means that it is safe to establish a physical connection to the LAN without any previous instrument configuration.

NOTICE

Risk of network errors

Connection errors can affect the entire network. If your network does not support DHCP, or if you choose to disable dynamic TCP/IP configuration, you must assign valid address information before connecting the instrument to the LAN. Contact your network administrator to obtain a valid IP address.

Assigning the IP address on the R&S FSW

1. Press the SETUP key.
2. Press the "Network + Remote" softkey.
3. Select the "Network" tab.
4. In the "Network + Remote" dialog, toggle the "DHCP On/Off" setting to the required mode.
   If DHCP is "Off", you must enter the IP address manually, as described in the following steps.

Note: When DHCP is changed from "On" to "Off", the previously set IP address and subnet mask are retrieved.
If DHCP is "On", the IP address of the DHCP server is obtained automatically. The configuration is saved, and you are prompted to restart the instrument. You can skip the remaining steps.
Note: When a DHCP server is used, a new IP address may be assigned each time the instrument is restarted. This address must first be determined on the instrument itself. Thus, when using a DHCP server, it is recommended that you use the permanent computer name, which determines the address via the DNS server.

5. Enter the "IP Address", for example *10.0.0.10*. The IP address consists of four number blocks separated by dots. Every block contains 3 numbers in maximum.
6. Enter the "Subnet Mask", for example *255.255.255.0*. The subnet mask consists of four number blocks separated by dots. Every block contains 3 numbers in maximum.
7. Close the dialog box.

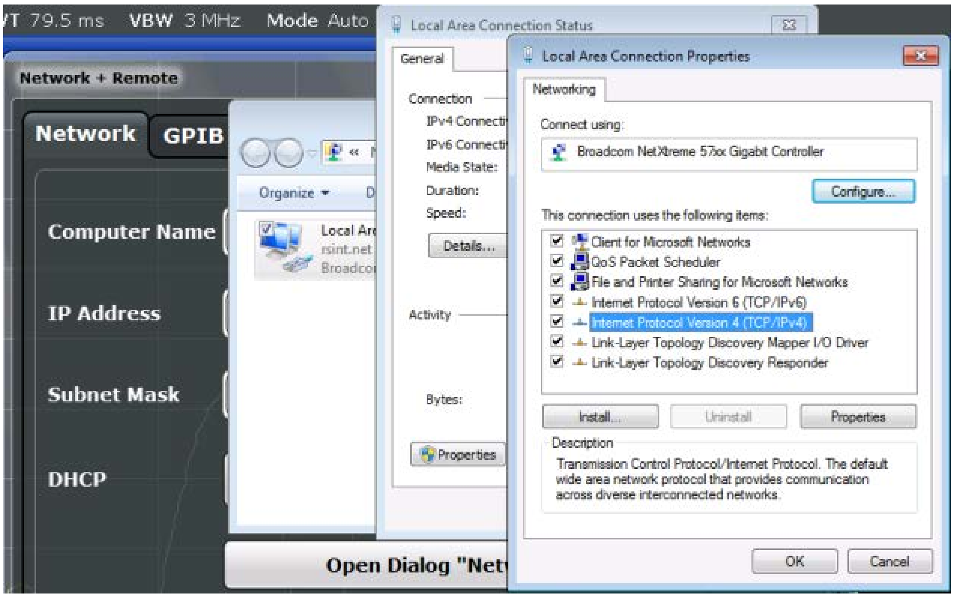
   If you have entered an invalid IP address or subnet mask, the message "out of range" is displayed in the status line. If the settings are correct, the configuration is saved, and you are prompted to restart the instrument.
8. Confirm the displayed message ("Yes" button) to restart the instrument.

Using a DNS server to determine the IP address

If a DNS server is configured on the R&S FSW, the server can determine the current IP address for the connection using the permanent computer name.

1. Obtain the name of your DNS domain and the IP addresses of the DNS and WINS servers on your network (see chapter 9.5.1.3, "How to Change the Instrument Name", on page 429).
2. Press the SETUP key and then the "Network + Remote" softkey.
3. In the "Network" tab, select the "Open Dialog 'Network Connections'" button.
4. Double-tap the "Local Area Network" icon.
5. In the "Local Area Connection Status" dialog box, select the "Properties" button.

   The items used by the LAN connection are displayed.
6. Tap the entry named "Internet Protocol Version 4 (TCP/IPv4)" to highlight it.



7. Select the "Properties" button.
8. On the "General" tab, select "Use the following DNS server addresses" and enter your own DNS addresses.

   For more information refer to the Windows operating system Help.

9.5.1.3 How to Change the Instrument Name

In a LAN that uses a DNS server (Domain Name System server), each PC or instrument connected in the LAN can be accessed via an unambiguous computer name instead of the IP address. The DNS server translates the host name to the IP address. This is especially useful when a DHCP server is used, as a new IP address may be assigned each time the instrument is restarted.

Each instrument is delivered with an assigned computer name, but this name can be changed.

To change the computer name

1. Press the SETUP key and then the "Network + Remote" softkey.
   The current "Computer Name" is displayed in the "Network" tab.
2. Enter the new computer name and close the dialog box.

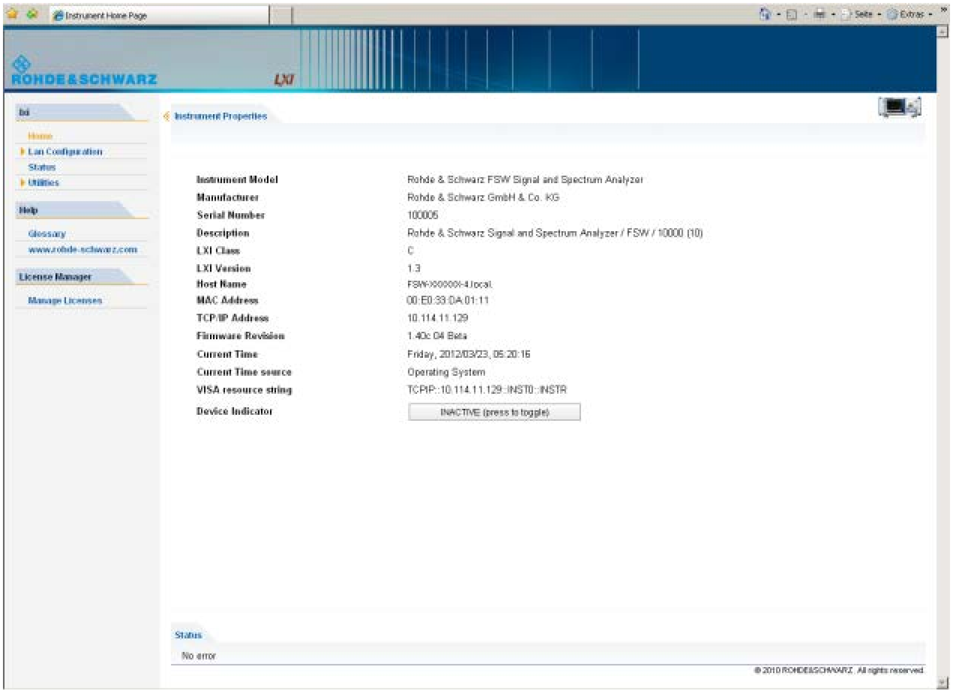
   The configuration is saved, and you are prompted to restart the instrument.
3. Confirm the displayed message ("Yes" button) to restart the instrument.

9.5.1.4 How to Configure the LAN Using the LXI Browser Interface

The instrument's LXI browser interface works correctly with all W3C compliant browsers.

► In the web browser, open the `http://<instrument-hostname>` or `http://<instrument-ip-address>` page, e.g. *http://10.113.10.203*.
The default password to change LAN configurations is *LxiWebIfc*.

The "Instrument Home Page" (welcome page) opens.



The instrument home page displays the device information required by the LXI standard including the VISA resource string in read-only format.

► Press the "Device Indicator" button on the "Instrument Home Page" to activate or deactivate the LXI status icon on the status bar of the R&S FSW. A green LXI status symbol indicates that a LAN connection has been established; a red symbol indicates an error, for example, that no LAN cable is connected. When a device is connecting to the instrument, the LXI logo blinks. The "Device Indicator" setting is not password-protected.

The most important control elements in the navigation pane of the browser interface are the following:

- "LAN Configuration" opens the menu with configuration pages.
- "Status" displays information about the LXI status of the instrument.
- "Help > Glossary" opens a document with a glossary of terms related to the LXI standard.

LAN Configuration

The LAN configuration consists of three parts:

- "IP configuration" provides all mandatory LAN parameters.

- "Advanced LAN Configuration" provides LAN settings that are not declared mandatory by the LXI standard.
- "Ping Client" provides the ping utility to verify the connection between the instrument and other devices.

IP Configuration

The "LAN Configuration > IP configuration" web page displays all mandatory LAN parameters and allows their modification.

The "TCP/IP Mode" configuration field controls how the IP address for the instrument gets assigned (see also chapter 9.5.1.2, "How to Assign the IP Address", on page 427). For the manual configuration mode, the static IP address, subnet mask, and default gateway are used to configure the LAN. The automatic configuration mode uses DHCP server or Dynamic Link Local Addressing (Automatic IP) to obtain the instrument IP address.

Changing the LAN configuration is password-protected. The default password is *LxiWebIfc* (notice upper and lower case characters).

You can change the LXI password in the "Network + Remote" dialog box, see chapter 9.4.4, "LXI Settings", on page 424.

Advanced LAN Configuration

The "LAN Configuration > Advanced LAN Configuration" parameters are used as follows:

- The "Negotiation" configuration field provides different Ethernet speed and duplex mode settings. In general, the "Auto Detect" mode is sufficient.
- "ICMP Ping" must be enabled to use the ping utility.
- "VXI-11" is the protocol that is used to detect the instrument in the LAN. According to the standard, LXI devices must use VXI-11 to provide a detection mechanism; other additional detection mechanisms are permitted.
- mDNS and DNS-SD are two additional protocols: Multicast DNS and DNS Service Discovery. They are used for device communication in zero configuration networks working without DNS and DHCP

Ping Client

Ping is a utility that verifies the connection between the LXI-compliant instrument and another device. The ping command uses the `ICMP` echo request and echo reply packets to determine whether the LAN connection is functional. Ping is useful for diagnosing IP network or router failures. The ping utility is not password-protected.

To initiate a ping between the LXI-compliant instrument and a second connected device:

1. Enable "ICMP Ping" on the "Advanced LAN Configuration" page (enabled after an LCI).
2. Enter the IP address of the second device **without the ping command and without any further parameters** into the "Destination Address" field (e.g. *10.113.10.203*).

3. Select "Submit".

9.5.1.5 How to Change the GPIB Instrument Address

In order to operate the instrument via remote control, it must be addressed using the GPIB address. The remote control address is factory-set to 20, but it can be changed if it does not fit in the network environment. For remote control, addresses 0 through 30 are allowed. The GPIB address is maintained after a reset of the instrument settings.

Setting the GPIB address

1. On the R&S FSW, press the SETUP key.
2. Press the "Network + Remote" softkey.
3. In the "Network + Remote" dialog box, select the "GPIB" tab.
4. In the "GPIB Address" field, enter a value between 0 and 30.

SCPI command:

```
SYST:COMM:GPIB:ADDR 18
```

9.5.2 How to Operate the Instrument Without a Network

To operate the instrument without a network connection either temporarily or permanently, no special measures are necessary. Windows automatically detects the interruption of the network connection and does not set up the connection when the instrument is switched on.

If you are not prompted to enter the user name and password, proceed as described in "Reactivating the automatic login mechanism" on page 434.

9.5.3 How to Log on to the Network

Windows requires that users identify themselves by entering a user name and password in a login window. You can set up two types of user accounts, either an administrator account with unrestricted access to the computer/domain or a standard user account with limited access. The instrument provides an auto-login function for the administrator account, i.e. login with unrestricted access is carried out automatically in the background. By default, the user name for the administrator account is "Instrument", and the user name for the standard user account is "NormalUser". In both cases the initial password is "894129". You can change the password in Windows for any user at any time. Some administrative tasks require administrator rights (e.g. firmware updates or the configuration of a LAN network). Refer to chapter 8, "General Instrument Setup", on page 341 to find out which functions are affected.

At the same time you log on to the operating system, you are automatically logged on to the network. As a prerequisite, the user name and the password must be identical on the instrument and on the network.

9.5.3.1 How to Create Users

After the software for the network has been installed, the instrument issues an error message the next time it is switched on because there is no user named "instrument" (= default user ID for Windows automatic login) in the network. Thus, a matching user must be created in the R&S FSW and in the network, the password must be adapted to the network password, and the automatic login mechanism must then be deactivated.

The network administrator is responsible for creating new users in the network. A new user can be created on the instrument using the "User Account" dialog box:

1. Select the "Windows" icon in the toolbar to access the operating system.
2. In the "Start" menu, select "Control Panel" and then select "User Accounts."
3. Select "Give other users access to this computer" and then "Add".
   The "Add New User" dialog box is displayed.
4. Enter the name of the new user in the text field and select "Next".
5. Define the level of access you want to allow the new user:
   - Select "Standard" to create an account with limited rights.
   - Select "Administrator" to create an account with administrator rights.

   Note: Full firmware functionality requires administrator rights.
6. Select the "Finish" button.
   The new user is created.

9.5.3.2 How to Change the User Password

After the new user has been created on the instrument, the password must be adapted to the network password. This is also done using the "User Accounts" dialog box.

1. Select the "Windows" icon in the toolbar to access the operating system.
2. In the "Start" menu, select "Control Panel" and then select "User Accounts".
3. Select "Manage User Accounts".
4. Select the desired user account and then "Reset Password...".
5. Enter the new password in the upper text line and repeat it in the following line.
6. Select the "Reset Password" button (at the end of the page).
   The new password is now active.

9.5.3.3 How to Activate or Deactivate the Automatic Login Mechanism

Deactivating the automatic login mechanism

When shipped, the instrument is already configured to automatically log on under Windows. To deactivate the automatic login mechanism, perform the following steps:

1. In the "Start" menu, select "Run".
   The "Run" dialog box is displayed.
2. Enter the command `C:\R_S\INSTR\USER\NO_AUTOLOGIN.REG`.
3. Press the ENTER key to confirm.
   The automatic login mechanism is deactivated. The next time you switch on the instrument, you are prompted to enter your user name and password before the firmware is started.

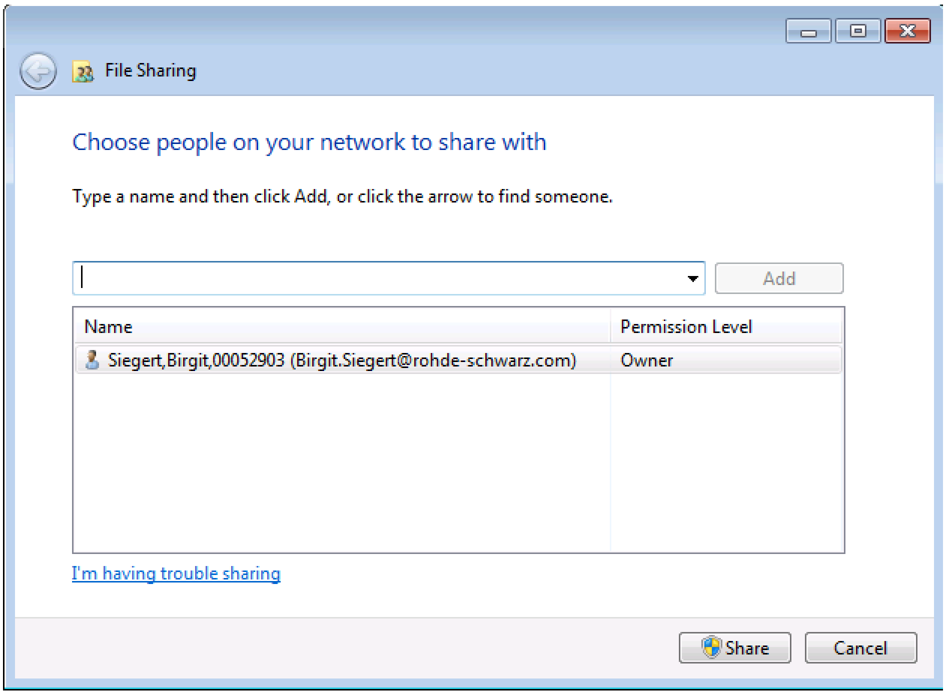
Reactivating the automatic login mechanism

1. In the "Start" menu, select "Run".
   The "Run" dialog box is displayed.
2. Enter the command `C:\R_S\INSTR\USER\AUTOLOGIN.REG`.
3. Press the ENTER key to confirm.
   The automatic login mechanism is reactivated. It will be applied the next time the instrument is switched on.

9.5.4 How to Share Directories (only with Microsoft Networks)

Sharing directories makes data available for other users. This is only possible in Microsoft networks. Sharing is a property of a file or directory.

1. In the "Start" menu, select "Programs", "Accessories" and then select "Windows Explorer".
2. Select the desired folder with the right mouse button.
3. In the context menu, select "Sharing with > Specific people".
   The dialog box for sharing a directory is displayed.



4. Select a user from the list or add a new name and select the "Add" button.
5. Select the "Share" button.
6. Select "Done" to close the dialog box.
   The drive is shared and can be accessed by the selected users.

9.5.5 How to Set Up Remote Desktop

Remote Desktop is a Windows application which can be used to access and control the instrument from a remote computer through a LAN connection. While the instrument is in operation, the instrument screen contents are displayed on the remote computer, and Remote Desktop provides access to all of the applications, files, and network resources of the instrument. Thus, remote operation of the R&S FSW is possible.

With Windows7, Remote Desktop Client is part of the operating system. For other versions of Windows, Microsoft offers the Remote Desktop Client as an add-on.

This section provides basic instructions on setting up the Remote Desktop for the R&S FSW. For details refer to the Windows operating system documentation.

9.5.5.1 How to Configure the R&S FSW for Remote Operation via Remote Desktop

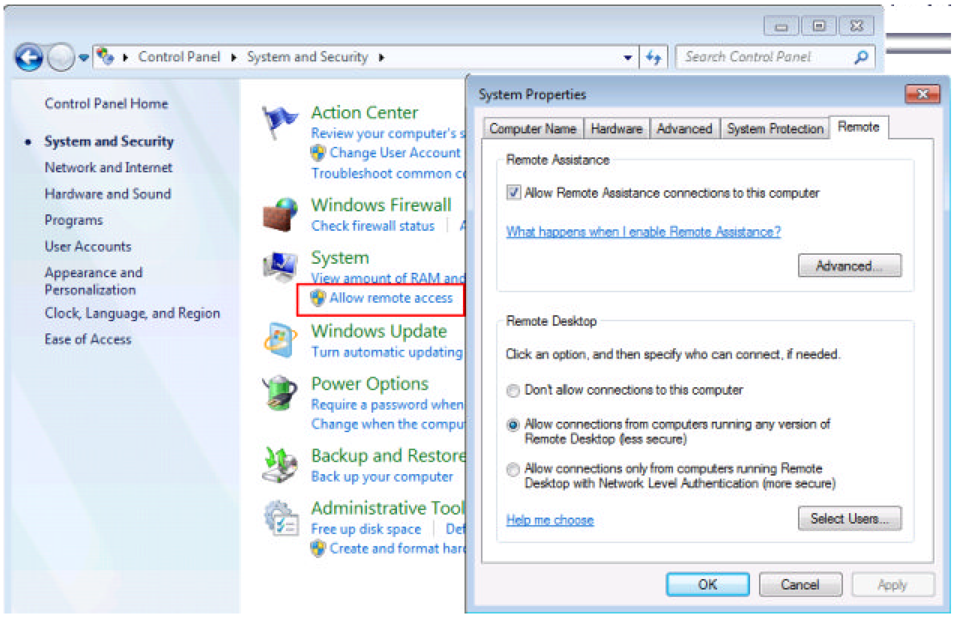
1. Create a fixed IP address for the TCP/IP protocol as described in "Assigning the IP address on the R&S FSW" on page 427.

   Note: To avoid problems, use a fixed IP address.

When a DHCP server is used, a new IP address is assigned each time the instrument is restarted. This address must first be determined on the instrument itself. Thus, using a DHCP server is not suitable for remote operation of the R&S FSW via Remote Desktop.

2. 

   Select the "Windows" icon in the toolbar to access the operating system.
3. In the "Start" menu, select the "Control Panel" and then "System and Security".
4. In the "System" area, select "Allow remote access".



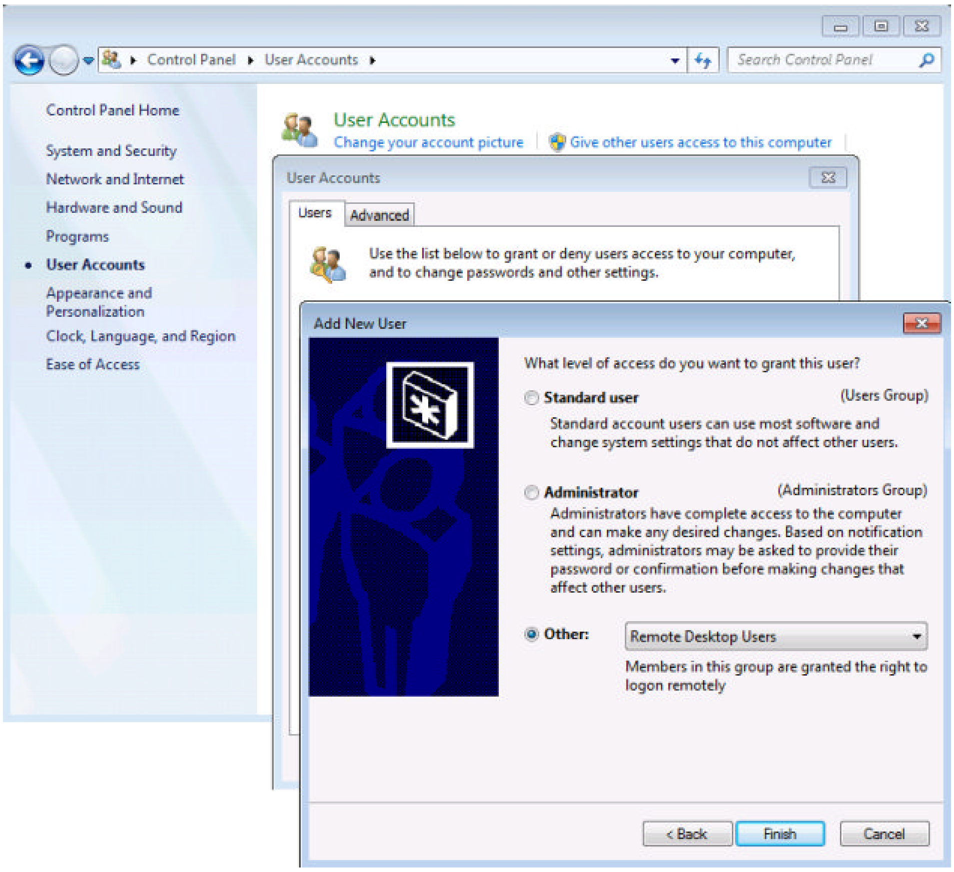
5. In the "Remote" tab, in the "Remote Desktop" area, activate the "Allow connections from computers running Remote Desktop" option.
6. Define which users are to be given access to the R&S FSW via Remote Desktop.
   Note: The user account under which configuration is carried out is automatically enabled for Remote Desktop.
   a) Select the "Select Users" button.
   b) Select the users or create new user accounts as described in chapter 9.5.3.1, "How to Create Users", on page 433.
   c) Select "OK" to confirm the settings.
7. The R&S FSW is now ready for connection setup with the Remote Desktop program of the controller.

9.5.5.2 How to Add or Remove Users to the Remote Desktop Users Group

Only users in the Remote Desktop Users Group are allowed to connect to the R&S FSW via Remote Desktop. You can add the users to this group directly when you allow remote access on the R&S FSW, as described in chapter 9.5.5, "How to Set Up Remote Desktop", on page 435. Furthermore, you can add or remove users to this group at any time.

1. Select the "Windows" icon in the toolbar to access the operating system.
2. In the "Start" menu, select "Control Panel" and then select "User Accounts."
3. Select "Give other users access to this computer".
   The "User Accounts" dialog box is displayed with a list of users and the user group they are assigned to.
4. Give existing users the right to access the R&S FSW via Remote Desktop:
   a) Select the user from the list and then select the "Properties" button.
   b) As the level of access you want to allow the user, select "Other", then select "Remote Desktop Users" from the dropdown list.
   c) Select the "Finish" button.
   The user is added to the Remote Desktop Users Group.
5. Create new users who may access the R&S FSW via Remote Desktop:
   a) Select "Add".
   b) Enter the name of the new remote user in the text field and select "Next".

c) As the level of access you want to allow the new user, select "Other", then select "Remote Desktop Users" from the dropdown list.



d) Select the "Finish" button.
The user is created.

6. Remove users from the Remote Desktop Users Group:

a) Select the user from the list and then select the "Remove" button.

b) Select "OK" to confirm the action.

Note: Users with administrator rights retain their access rights via Remote Desktop even when they have been removed from the Remote Desktop Users Group.

7. Select "OK" to close the dialog box.

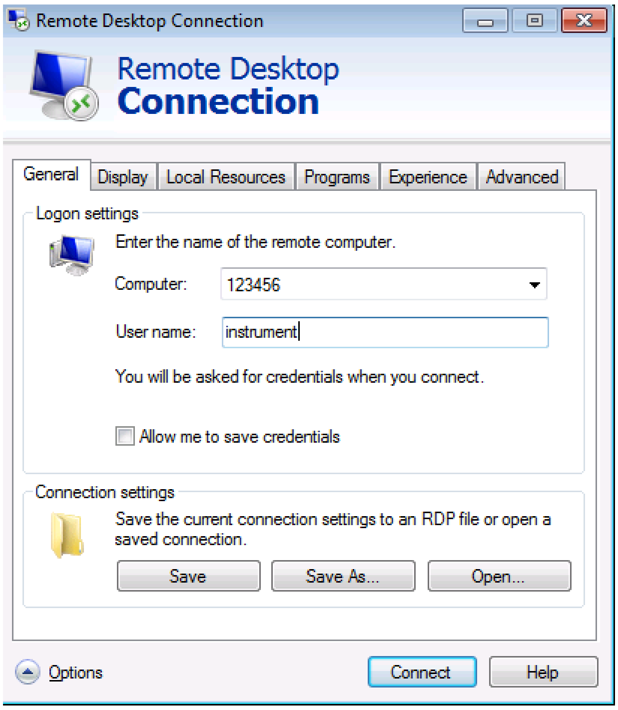
9.5.5.3 How to Configure the Controller

Remote Desktop Client

With Windows7, Remote Desktop Client is part of the operating system and can be accessed via "Start > Programs > Accessories > Communications > Remote Desktop Connection".

For other versions of Windows, Microsoft offers the Remote Desktop Client as an add-on.

1. In the "Start" menu of the controller, select "All Programs > Accessories > Remote Desktop Connection".
   The "Remote Desktop Connection" dialog box is displayed.
2. Select the "Options >>" button.
   The dialog box is expanded to display the configuration data.



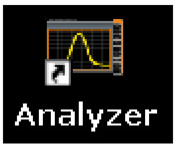
3. Open the "Experience" tab.
   The settings on this tab are used to select and optimize the connection speed.
4. In the list, select the appropriate connection (for example: LAN (10 Mbps or higher)).
   Depending on your selection (and how powerful the connection is), the options are activated or deactivated.
5. To improve the performance, you can deactivate the "Desktop background", "Show window contents while dragging" and "Menu and window animation" options.
6. Open the "Local Resources" tab to enable printers, local drives and serial interfaces.
7. If you will need to access drives of the controller from the R&S FSW (e.g. in order to store settings or to copy files from the controller to the R&S FSW), activate the "Disk drives" option.
   Windows will then map drives of the controller to the corresponding network drives.
8. To use printers connected to the controller while accessing them from the R&S FSW, activate the "Printers" options. Do not change the remaining settings.
9. Open the "Display" tab.
   The options to configure the R&S FSW screen display are displayed.

10. Under "Remote desktop size", you can set the size of the R&S FSW window on the desktop of the controller.
11. Under "Colors", do not change the settings.
12. Set the "Display the connection bar when I use the full screen" option:
    If activated, a bar showing the network address of the R&S FSW will appear at the top edge of the screen. You can use this bar to reduce, minimize or close the window.
    If deactivated, the only way you can return to the controller desktop from the R&S FSW screen in full screen mode is to select "Disconnect" from the "Start" menu.

9.5.5.4 How to Start and Close the Remote Desktop

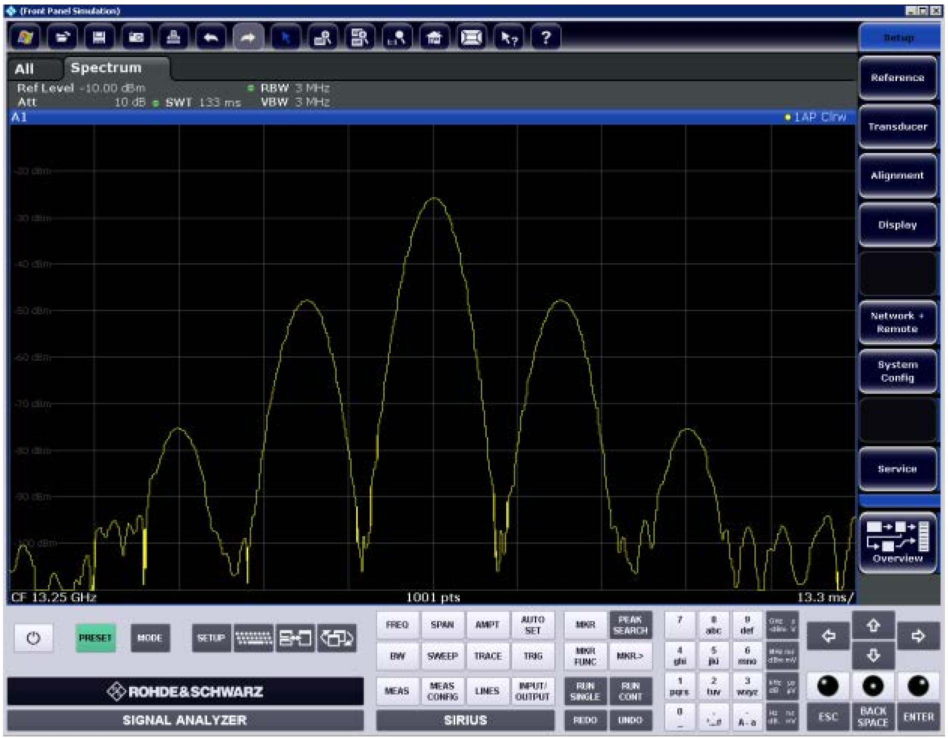
To set up a connection to the R&S FSW

1. In the "Remote Desktop Connection" dialog box (see chapter 9.5.5.3, "How to Configure the Controller", on page 438), open the "General" tab.
2. In the "Computer" field, enter the IP address of the R&S FSW.
   In the "User name" field, enter *instrument* to log in as an administrator, or *Normal User* to log in as a standard user.
   In the "Password" field, enter *894129*.
3. To save the connection configuration for later use:
   a) Select the "Save As" button.
      The "Save As" dialog box is displayed.
   b) Enter the name for the connection information (`*.RDP`).
4. To load an existing connection configuration:
   a) Select the "Open" button.
      The "Open" dialog box is displayed.
   b) Select the `*.RDP` file.
5. Select the "Connect" button.
   The connection is set up.
6. If the "Disk drives" option is activated on the "Local Resources" tab, a warning is displayed indicating that the drives are enabled for access from the R&S FSW.
   Select "OK" to confirm the warning.
7. After a few moments, the R&S FSW screen is displayed.
   If a dark screen appears or a dark square appears in the upper left-hand corner of the screen, you must restart the R&S FSW in order to see the modified screen resolution.

	• Press the key combination ALT + F4. • The R&S FSW firmware is shut down, which may take a few seconds. • On the desktop, double-tap the "Analyzer" icon.

The firmware restarts and then automatically opens the "Soft Front Panel", i.e. the user interface on which all front panel controls and the rotary knob are mapped to buttons. For details see chapter 8.6.7, "How to Work with the Soft Front Panels", on page 379.

8. To deactivate or activate the "Softfrontpanel", press the F6 key.
   After the connection is established, the R&S FSW screen is displayed in the "Remote Desktop" application window.



The Windows "Start" menu can be made available by expanding the "Remote Desktop" window to full size.
During the connection with the controller, the login entry is displayed on the R&S FSW screen.

To terminate Remote Desktop control

The connection can be terminated by the controller or by a user at the R&S FSW:

1. On the controller, close the "Remote Desktop" window at any time.
   The connection to the R&S FSW is terminated.
2. On the R&S FSW, a user logs on.
   The connection to the controller is terminated as a result. A message is displayed on the controller display indicating that another user has assumed control of the instrument.

Restoring the connection to the R&S FSW

Follow the instructions above for setting up a connection to the R&S FSW. If the connection is terminated and then restored, the R&S FSW remains in the same state.

9.5.5.5 How to Shut Down the R&S FSW via Remote Operation

1. Select the R&S FSW softfrontpanel and close the application with the key combination ALT + F4.
2. Select the desktop and press the key combination ALT + F4.
   A safety query is displayed to warn you that the instrument cannot be reactivated via remote operation and asks you whether you want to continue the shutdown process.
3. Respond to the safety query with "Yes".
   The connection with the controller is terminated and the R&S FSW is shut down.

9.5.6 How to Start a Remote Control Session from a PC

When you switch on the instrument, it is always in manual operation state ("local" state) and can be operated via the front panel.

To start remote control

1. Send an addressed command (`GTR` - Go to Remote) from a controller to the instrument.

   The instrument is switched to remote control ("remote" state). Operation via the front panel is disabled. Only the "Local" softkey is displayed to return to manual operation. The instrument remains in the remote state until it is reset to the manual state via the instrument or via remote control interfaces. Switching from manual operation to remote control and vice versa does not affect the other instrument settings.
2. During program execution, send the `SYSTem:DISPlay:UPDate ON` command to activate the display of results (see `SYSTem:DISPlay:UPDate` on page 716).

   The changes in the device settings and the recorded measurement values are displayed on the instrument screen.
3. To obtain optimum performance during remote control, send the `SYSTem:DISPlay:UPDate OFF` command to hide the display of results and diagrams again (default setting in remote control).
4. To prevent unintentional return to manual operation, disable the keys of the instrument using the universal command `LLO`.

   Switching to manual mode is only possible via remote control then. This function is only available for the GPIB interface.
5. To enable the keys of the R&S FSW again, switch the instrument to local mode (`GTL` - Go to Local), i.e. deactivate the `REN` line of the remote control interface.

If the instrument is operated exclusively in remote control, it is recommended that you switch on the power-save mode for the display. For more details on this mode refer to the "Overview".

9.5.7 How to Return to Manual Operation

Before you switch back to manual operation, all remote command processing must be completed. Otherwise, the instrument will switch back to remote control immediately.

► Select the "Local" softkey or the PRESET key, or use the following GPIB command:
`status = viGpibControlREN(vi, VI_GPIB_REN_ADDRESS_GTL)`

10 Remote Commands

The commands required to perform measurements in the Spectrum application in a remote environment are described here. It is assumed that the R&S FSW has already been set up for remote operation in a network as described in chapter 9.5, "How to Set Up a Network and Remote Control", on page 425.

Common Suffixes

In the Spectrum application, the following common suffixes are used in remote commands:

Suffix	Value range	Description
<ch>	1..18 (TX channel) 1..11 (ALT channel)	Channel
<k>	1..8 (Limit line) 1 | 2 (Display line)	Line
<m>	1..16	Marker
<n>	1..16	Window
<t>	1..6	Trace

Compatibility with former R&S signal and spectrum analyzers

As a rule, the R&S FSW supports most commands from previous R&S signal and spectrum analyzers such as the FSQ, FSP, FSU, or FSV. However, the default values, in particular the number of sweep points or particular bandwidths, may vary. Therefore, the R&S FSW can emulate these other devices, including their default values, in order to repeat previous measurements or support existing control applications as in legacy systems.

10.1 Common Commands

Common commands are described in the IEEE 488.2 (IEC 625-2) standard. These commands have the same effect and are employed in the same way on different devices. The headers of these commands consist of "*" followed by three letters. Many common commands are related to the Status Reporting System.

Available common commands:

***CAL?**

Calibration Query

Initiates a calibration of the instrument and subsequently queries the calibration status. Responses > 0 indicate errors.

Usage: Query only

Manual control: See "Starting a Self-alignment" on page 352

***CLS**

CLear Status

Sets the status byte (STB), the standard event register (ESR) and the `EVENt` part of the `QUEStionable` and the `OPERation` registers to zero. The command does not alter the mask and transition parts of the registers. It clears the output buffer.

Usage: Setting only

***ESE** <Value>

Event Status Enable

Sets the event status enable register to the specified value. The query returns the contents of the event status enable register in decimal form.

Parameters:
<Value> Range: 0 to 255

***ESR?**

Event Status Read

Returns the contents of the event status register in decimal form and subsequently sets the register to zero.

Return values:
<Contents> Range: 0 to 255

Usage: Query only

***IDN?**

IDeNtification: returns the instrument identification.

Return values:
<ID> "Rohde&Schwarz,<device type>,<serial number>,<firmware version>"

Example: `Rohde&Schwarz,FSW-26,1312.8000K26/100005,1.30`

Usage: Query only

Manual control: See "*IDN Format" on page 420

***IST?**

Individual STatus query

Returns the contents of the IST flag in decimal form. The IST flag is the status bit which is sent during a parallel poll.

Return values:
<ISTflag> 0 | 1

Usage: Query only

***OPC**

OPeration Complete

Sets bit 0 in the event status register when all preceding commands have been executed. This bit can be used to initiate a service request. The query form writes a "1" into the output buffer as soon as all preceding commands have been executed. This is used for command synchronization.

***OPT?**

OPTion identification query

Queries the options included in the instrument. For a list of all available options and their description refer to the CD-ROM.

Return values:

<Options>	The query returns a list of all installed and activated options, separated by commas, where: B<number> describes hardware options K<number> describes software options For PSA89600 emulation, the option is indicated as "B7J" for the `*OPT?` query ("B7J, 140" if `SYST:PSA:WIDeband` is activated).
Usage:	Query only

***PCB** <Address>

Pass Control Back

Indicates the controller address to which remote control is returned after termination of the triggered action.

Setting parameters:

<Address>	Range:	0 to 30
Usage:	Setting only	

***PRE** <Value>

Parallel poll Register Enable

Sets parallel poll enable register to the indicated value. The query returns the contents of the parallel poll enable register in decimal form.

Parameters:

<Value>	Range:	0 to 255

***PSC** <Action>

Power on Status Clear

Determines whether the contents of the `ENABle` registers are preserved or reset when the instrument is switched on. Thus a service request can be triggered when the instrument is switched on, if the status registers ESE and SRE are suitably configured. The query reads out the contents of the "power-on-status-clear" flag.

Parameters:

<Action>	0 | 1 **0** The contents of the status registers are preserved. **1** Resets the status registers.

***RST**

ReSeT

Sets the instrument to a defined default status. The default settings are indicated in the description of commands.

The command is equivalent to `SYSTem:PRESet`.

Note that the factory set default values can be modified to user-defined settings (see `MMEMory:LOAD:STATe` on page 688). For more details on default values see chapter 7.1, "Restoring the Default Instrument Configuration (Preset)", on page 318.

Usage: Setting only

***SRE** <Contents>

Service Request Enable

Sets the service request enable register to the indicated value. This command determines under which conditions a service request is triggered.

Parameters:

<Contents>	Contents of the service request enable register in decimal form. Bit 6 (MSS mask bit) is always 0. Range: 0 to 255

***STB?**

STatus Byte query

Reads the contents of the status byte in decimal form.

Usage: Query only

***TRG**

TRiGger

Triggers all actions waiting for a trigger event. In particular, *TRG generates a manual trigger signal (Manual Trigger). This common command complements the commands of the `TRIGger` subsystem.

`*TRG` corresponds to the `INITiate:IMMediate` command (see `INITiate[:IMMediate]` on page 461).

Usage: Event

***TST?**

self TeST query

Triggers selftests of the instrument and returns an error code in decimal form (see Service Manual supplied with the instrument). "0" indicates no errors occured.

Usage: Query only

***WAI**

WAIt to continue

Prevents servicing of the subsequent commands until all preceding commands have been executed and all signals have settled (see also command synchronization and `*OPC`).

Usage: Event

10.2 Selecting the Operating Mode and Application

The following commands are required to select the operating mode or the application and to configure a Sequencer in a remote environment. The tasks for manual operation are described in chapter 3, "Applications and Operating Modes", on page 18.

10.2.1 Selecting the Mode and Applications

DISPlay:ATAB <State>

This command switches between the MultiView tab and the most recently displayed measurement channel.

Parameters:

<State> ON | OFF

ON

The MultiView tab is displayed.

OFF

The most recently displayed measurement channel is displayed.

*RST: OFF

INSTrument:CREate[:NEW] <ChannelType>, <ChannelName>

This command adds an additional measurement channel. The number of measurement channels you can configure at the same time depends on available memory.

See also

- `INSTrument[:SELect]` on page 453
- `INSTrument:DELete` on page 451

Parameters:

<ChannelType> Channel type of the new channel.
For a list of available channel types see table 10-1.

<ChannelName> String containing the name of the channel. The channel name is displayed as the tab label for the measurement channel.
Note: If the specified name for a new channel already exists, the default name, extended by a sequential number, is used for the new channel (see table 10-1).

Example: `INST:CRE SAN, 'Spectrum 2'`
Adds an additional spectrum display named "Spectrum 2".

Manual control: See "New Channel" on page 25

INSTrument:CREate:REPLace <ChannelName1>,<ChannelType>,<ChannelName2>

This command replaces a measurement channel with another one.

Parameters:

<ChannelName1> String containing the name of the measurement channel you want to replace.

<ChannelType> Channel type of the new channel.
For a list of available channel types see table 10-1.

<ChannelName2> String containing the name of the new channel.
Note: If the specified name for a new channel already exists, the default name, extended by a sequential number, is used for the new channel (see table 10-1).

Example: `INST:CRE:REPL 'Spectrum2',IQ,'IQAnalyzer'`
Replaces the channel named 'Spectrum2' by a new measurement channel of type 'IQ Analyzer' named 'IQAnalyzer'.

Manual control: See "Replace Current Channel" on page 25

INSTrument:DELete <ChannelName>

This command deletes a measurement channel. If you delete the last measurement channel, the default "Spectrum" channel is activated.

Parameters:

<ChannelName> String containing the name of the channel you want to delete.
A measurement channel must exist in order to be able delete it.

Example: `INST:DEL 'Spectrum4'`
Deletes the spectrum channel with the name 'Spectrum4'.

INSTrument:LIST?

This command queries all active measurement channels. This is useful in order to obtain the names of the existing measurement channels, which are required in order to replace or delete the channels.

Return values:

<ChannelType>, <ChannelName> For each channel, the command returns the channel type and channel name (see table 10-1).
Tip: to change the channel name, use the `INSTrument:REName` command.

Example: `INST:LIST?`
Result for 3 measurement channels:
`'ADEM','Analog Demod','IQ','IQ Analyzer','SANALYZER','Spectrum'`

Usage: Query only

Table 10-1: Available measurement channel types and default channel names

Application	<ChannelType> Parameter	Default Channel Name*)
Spectrum	SANALYZER	Spectrum
I/Q Analyzer	IQ	IQ Analyzer
Pulse (R&S FSW-K6)	PULSE	Pulse
Analog Demodulation (R&S FSW-K7)	ADEM	Analog Demod
GSM (R&S FSW-K10)	GSM	GSM
Multi-Carrier Group Delay (R&S FSW-K17)	MCGD	MC Group Delay
Noise (R&S FSW-K30)	NOISE	Noise
Phase Noise (R&S FSW-K40)	PNOISE	Phase Noise
Note: the default channel name is also listed in the table. If the specified name for a new channel already exists, the default name, extended by a sequential number, is used for the new channel.		

Application	**<ChannelType> Parameter**	**Default Channel Name*)**
VSA (R&S FSW-K70)	DDEM	VSA
3GPP FDD BTS (R&S FSW-K72)	BWCD	3G FDD BTS
3GPP FDD UE (R&S FSW-K73)	MWCD	3G FDD UE
cdma2000 BTS (R&S FSW-K82)	BC2K	CDMA2000 BTS
cdma2000 MS (R&S FSW-K83)	MC2K	CDMA2000 MS
1xEV-DO BTS (R&S FSW-K84)	BDO	1xEV-DO BTS
1xEV-DO MS (R&S FSW-K85)	MDO	1xEV-DO MS
WLAN (R&S FSW-K91)	WLAN	WLAN
LTE (R&S FSW-K10x)	LTE	LTE
Note: the default channel name is also listed in the table. If the specified name for a new channel already exists, the default name, extended by a sequential number, is used for the new channel.		

INSTrument:MODE <OpMode>

The operating mode of the R&S FSW determines which applications are available and active. Whenever you change the operating mode, the currently active measurement channels are stored. The default operating mode is Signal and Spectrum Analyzer mode, however, the presetting can be changed.

For details on operating modes and applications see chapter 3, "Applications and Operating Modes", on page 18.

Parameters:

<OpMode> SANalyzer | MSRanalyzer

Example: `INST:MODE MSR`
Switches to MSRA mode.

Usage: SCPI confirmed

INSTrument:REName <ChannelName1>, <ChannelName2>

This command renames a measurement channel.

Parameters:

<ChannelName1> String containing the name of the channel you want to rename.

<ChannelName2> String containing the new channel name.
Note that you can not assign an existing channel name to a new channel; this will cause an error.

Example: `INST:REN 'Spectrum2','Spectrum3'`
Renames the channel with the name 'Spectrum2' to 'Spectrum3'.

INSTrument[:SELect] <ChannelType> | <ChannelName>

This command activates a new measurement channel with the defined channel type, or selects an existing measurement channel with the specified name.

Also see

- `INSTrument:CREate[:NEW]` on page 450
- chapter 10.2.3, "Programming Example: Performing a Sequence of Measurements", on page 456

Parameters:

<ChannelType> Channel type of the new channel.
For a list of available channel types see table 10-1.

<ChannelName> String containing the name of the channel.

Example: `INST SAN`
Activates a measurement channel for the Spectrum application.
`INST 'MySpectrum'`
Selects the measurement channel named 'MySpectrum' (for example before executing further commands for that channel).

Usage: SCPI confirmed

Manual control:
See "Spectrum" on page 20
See "1xEV-DO BTS" on page 20
See "1xEV-DO MS" on page 20
See "3G FDD BTS" on page 21
See "3G FDD UE" on page 21
See "Analog Demodulation" on page 21
See "cdma2000 BTS" on page 21
See "cdma2000 MS" on page 21
See "(Multi-Carrier) Group Delay" on page 22
See "GSM" on page 22
See "I/Q Analyzer" on page 22
See "LTE" on page 22
See "Noise Figure" on page 22
See "Phase Noise" on page 22
See "Pulse Measurements" on page 23
See "Vector Signal Analysis (VSA)" on page 23
See "WLAN" on page 23
See "New Channel" on page 25

10.2.2 Performing a Sequence of Measurements

The following commands control the sequencer.

For details on the Sequencer see chapter 3.5.1, "The Sequencer Concept", on page 26.

INITiate:SEQuencer:ABORt

This command stops the currently active sequence of measurements. The Sequencer itself is not deactivated, so you can start a new sequence immediately using `INITiate:SEQuencer:IMMediate` on page 454.

To deactivate the Sequencer use `SYSTem:SEQuencer` on page 455.

Usage: Event

Manual control: See "Sequencer State" on page 28

INITiate:SEQuencer:IMMediate

This command starts a new sequence of measurements by the Sequencer. Its effect is similar to the `INITiate[:IMMediate]` command used for a single measurement.

Before this command can be executed, the Sequencer must be activated (see `SYSTem:SEQuencer` on page 455).

Example:
`SYST:SEQ ON`
Activates the Sequencer.
`INIT:SEQ:MODE SING`
Sets single Sequencer mode so each active measurement will be performed once.
`INIT:SEQ:IMM`
Starts the sequential measurements.

Usage: Event

Manual control: See "Sequencer State" on page 28

INITiate:SEQuencer:MODE <Mode>

This command selects the way the R&S FSW application performs measurements sequentially.

Before this command can be executed, the Sequencer must be activated (see `SYSTem:SEQuencer` on page 455).

A detailed programming example is provided in chapter 10.2.3, "Programming Example: Performing a Sequence of Measurements", on page 456.

Note: In order to synchronize to the end of a sequential measurement using *OPC, *OPC? or *WAI you must use `SINGle` Sequencer mode.

For details on synchronization see chapter 9.1.6, "Command Sequence and Synchronization", on page 396

Parameters:

<Mode>

SINGle
Each measurement is performed once (regardless of the channel's sweep mode), considering each channels' sweep count, until all measurements in all active channels have been performed.

CONTinuous
The measurements in each active channel are performed one after the other, repeatedly (regardless of the channel's sweep mode), in the same order, until the Sequencer is stopped.

CDEFined
First, a single sequence is performed. Then, only those channels in continuous sweep mode (`INIT:CONT ON`) are repeated.

*RST: CONTinuous

Example:

`SYST:SEQ ON`
Activates the Sequencer.
`INIT:SEQ:MODE SING`
Sets single Sequencer mode so each active measurement will be performed once.
`INIT:SEQ:IMM`
Starts the sequential measurements.

Manual control: See "Sequencer Mode" on page 28

SYSTem:SEQuencer <State>

This command turns the Sequencer on and off. The Sequencer must be active before any other Sequencer commands (`INIT:SEQ...`) are executed, otherwise an error will occur.

A detailed programming example is provided in chapter 10.2.3, "Programming Example: Performing a Sequence of Measurements", on page 456.

Parameters:

<State>

ON | OFF

ON
The Sequencer is activated and a sequential measurement is started immediately.

OFF
The Sequencer is deactivated. Any running sequential measurements are stopped. Further Sequencer commands (`INIT:SEQ...`) are not available.

*RST: OFF

Example:
`SYST:SEQ ON`
Activates the Sequencer.
`INIT:SEQ:MODE SING`
Sets single Sequencer mode so each active measurement will be performed once.
`INIT:SEQ:IMM`
Starts the sequential measurements.
`SYST:SEQ OFF`

Manual control:
See "Sequencer State" on page 28

10.2.3 Programming Example: Performing a Sequence of Measurements

This example demonstrates how to perform several measurements in a sequence in a remote environment.

```
//2xSpectrumanalyzer + 2xIQ, start Sequencer at the end, test OPC?
// ------------------------------------------------------------------------


//------Preparing the instrument and first channel -----------
*RST
//Activate new IQ channel
INSTrument:CREate:NEW IQ,'IQ 1'
//Set sweep count for new IQ channel
SENS:SWEEP:COUNT 6
//Change trace modes for IQ channel
DISP:TRAC1:MODE BLANK
DISP:TRAC2:MODE MAXH
DISP:TRAC3:MODE MINH
//Switch to single sweep mode
INIT:CONT OFF
//switch back to first (default) analyzer channel
INST:SEL 'Spectrum';*WAI
//Switch into SEM
SENSe:SWEep:MODE ESPectrum
//Load Sem standard file for WCDMA
SENSe:ESPectrum:PRESet:STANdard 'WCDMA\3GPP\DL\3GPP_DL.xml'
//Set sweep count in Spectrum channel
SENS:SWEEP:COUNT 5

//----------Creating a second measurement channel ----------------

//Create second IQ channel
INSTrument:CREate:NEW IQ,'IQ 2'
//Set sweep count
SENS:SWEEP:COUNT 2
//Change trace modes
DISP:TRAC1:MODE MAXH
```

```
DISP:TRAC2:MODE MINH
//Create new analyzer channel
INSTrument:CREate:NEW SANalyzer,'Spectrum 2'
//Activate ACLR measurement in channel 'Spectrum 2'
CALCulate:MARKer:FUNCtion:POWer:SELect ACPower
//Load WCDMA Standard
CALCulate:MARKer:FUNCtion:POWer:PRESet FW3Gppcdma
//Change trace modes
DISP:TRAC2:MODE MAXH
DISP:TRAC1:MODE MINH

//--------Performing a sweep and retrieving results----------------

//Change sweep count
SENS:SWEep:COUNt 7
//Single Sweep mode
INIT:CONT OFF
//Switch back to first IQ channel
INST:SEL 'IQ 1';*WAI
//Perform a measurement
INIT:IMM;*OPC?
//Retrieve results
CALC:MARK:Y?
//Activate Multiview
DISPlay:ATAB     ON

//---------Performing a sequence of measurements with the Sequencer------------
//Activate Sequencer
SYSTem:SEQuencer ON
//Start sweep in Sequencer
INITiate:SEQuencer:IMMediate;*OPC?
//Switch into first IQ channel to get results
INST:SEL 'IQ 1';*WAI
CALCulate:MARKer:MAXimum
CALC:MARK:Y?
//Change sweep time in IQ
SENS:SWE:TIME 300us
//Switch to single Sequencer mode
INITiate:SEQuencer:MODE SINGle
//Sweep all channels once, taking the sweep count in each channel into account
INITiate:SEQuencer:IMMediate;*OPC?
//Set marker to maximum in IQ1 and query result
CALCulate:MARKer:MAXimum
CALC:MARK:Y?
//Switch to second IQ channel and retrieve results
INST:SEL 'IQ 2';*WAI
CALCulate:MARKer:MIN
CALC:MARK:Y?
//Switch to first Spectrum channel
INST:SEL 'Spectrum';*WAI
```

```
//Query one of the SEM results
CALCulate:MARKer:FUNCtion:POWer:RESult? CPOWer
//Switch to second Spectrum channel
INST:SEL 'Spectrum 2';*WAI
//Query channel power result
CALCulate:MARKer:FUNCtion:POWer:RESult? ACPower
```

10.3 Configuring and Performing Measurements

The following commands are required to configure measurements in a remote environment. The tasks for manual operation are described in chapter 4, "Measurements", on page 30.

10.3.1 Performing Measurements

10.3.1.1 Commands

Useful commands for performing measurements described elsewhere

- INITiate:ESPectrum on page 487
- INITiate:SPURious on page 511

Remote commands exclusive for performing measurements

ABORt

This command aborts a current measurement and resets the trigger system.

To prevent overlapping execution of the subsequent command before the measurement has been aborted successfully, use the `*OPC?` or `*WAI` command after `ABOR` and before the next command.

For details see chapter 9.1.6.1, "Preventing Overlapping Execution", on page 397.

To abort a sequence of measurements by the Sequencer, use the `INITiate:SEQuencer:ABORt` on page 454 command.

Note on blocked remote control programs:

If a sequential command cannot be completed, for example because a triggered sweep never receives a trigger, the remote control program will never finish and the remote channel (GPIB, LAN or other interface) to the R&S FSW is blocked for further commands. In this case, you must interrupt processing on the remote channel first in order to abort the measurement.

To do so, send a "Device Clear" command from the control instrument to the R&S FSW on a parallel channel to clear all currently active remote channels. Depending on the used interface and protocol, send the following commands:

- **Visa**: `viClear()`
- **GPIB**: `ibclr()`
- **RSIB**: `RSDLLibclr()`

Now you can send the `ABORt` command on the remote channel performing the measurement.

Example: `ABOR;:INIT:IMM`
Aborts the current measurement and immediately starts a new one.

Example: `ABOR;*WAI`
`INIT:IMM`
Aborts the current measurement and starts a new one once abortion has been completed.

Usage: SCPI confirmed

Manual control: See "Aborting the Self-alignment" on page 352

INITiate:CONMeas

This command restarts a (single) measurement that has been stopped (using `INIT:CONT OFF`) or finished in single sweep mode.

The measurement is restarted at the beginning, not where the previous measurement was stopped.

As opposed to `INITiate[:IMMediate]`, this command does not reset traces in maxhold, minhold or average mode. Therefore it can be used to continue measurements using maxhold or averaging functions.

Example: (for Spectrum application:)
`INIT:CONT OFF`
Switches to single sweep mode.
`DISP:WIND:TRAC:MODE AVER`
Switches on trace averaging.
`SWE:COUN 20`
Setting the sweep counter to 20 sweeps.
`INIT;*WAI`
Starts the measurement and waits for the end of the 20 sweeps.
`INIT:CONM;*WAI`
Continues the measurement (next 20 sweeps) and waits for the end.
Result: Averaging is performed over 40 sweeps.

Manual control: See "Continue Single Sweep" on page 205

INITiate:CONTinuous <State>

This command controls the sweep mode.

Note that in single sweep mode, you can synchronize to the end of the measurement with *OPC, *OPC? or *WAI. In continuous sweep mode, synchronization to the end of the measurement is not possible. Thus, it is not recommended that you use continuous sweep mode in remote control, as results like trace data or markers are only valid after a single sweep end synchronization.

For details on synchronization see chapter 9.1.6, "Command Sequence and Synchronization", on page 396

If the sweep mode is changed for a measurement channel while the Sequencer is active (see `INITiate:SEQuencer:IMMediate` on page 454) the mode is only considered the next time the measurement in that channel is activated by the Sequencer.

Parameters:
<State> ON | OFF

ON
Continuous sweep

OFF
Single sweep

*RST: ON

Example: `INIT:CONT OFF`
Switches the sweep mode to single sweep.
`INIT:CONT ON`
Switches the sweep mode to continuous sweep.

Manual control: See "Frequency Sweep" on page 31
See "Zero Span" on page 31
See "Continuous Sweep/RUN CONT" on page 204

INITiate[:IMMediate]

This command starts a (single) new measurement.

With sweep count or average count > 0, this means a restart of the corresponding number of measurements. With trace mode MAXHold, MINHold and AVERage, the previous results are reset on restarting the measurement.

You can synchronize to the end of the measurement with *OPC, *OPC? or *WAI.

For details on synchronization see chapter 9.1.6, "Command Sequence and Synchronization", on page 396

Example: (For Spectrum application:)
`INIT:CONT OFF`
Switches to single sweep mode.
`DISP:WIND:TRAC:MODE AVER`
Switches on trace averaging.
`SWE:COUN 20`
Sets the sweep counter to 20 sweeps.
`INIT;*WAI`
Starts the measurement and waits for the end of the 20 sweeps.

Manual control: See "Frequency Sweep" on page 31
See "Zero Span" on page 31
See "Single Sweep/ RUN SINGLE" on page 204

10.3.1.2 Programming Example: Performing a Basic Frequency Sweep

This example demonstrates how to configure and perform a basic frequency sweep measurement in a remote environment.

```
//--------------Preparing the measurement ---------------------
*RST
//Resets the instrument
INIT:CONT OFF
//Selects single sweep mode.

//--------------Configuring the Frequency and Span-------------
FREQ:CENT 100MHz
//Defines the center frequency
FREQ:SPAN 200MHz
//Sets the span to 100 MHz on either side of the center frequency.

//Activate signal tracking to keep the center frequency on the signal peak:
CALC:MARK:FUNC:STR ON
CALC:MARK:FUNC:STR:BAND 20MHz
CALC:MARK:FUNC:STR:THR -90dBm
```

```
CALC:MARK:FUNC:STR:TRAC 1
//After each sweep the maximum on trace 1 is searched within a range of 20 MHz
//around the center frequency. It must have a minimum power of -90dBm.

//--------------Configuring the Bandwidth----------------------
BAND:AUTO OFF
BAND 1MHz
BAND:TYPE RRC
//Defines the RBW as 1 MHz using an RRC filter

BAND:VID:AUTO OFF
BAND:VID 500kHz
//Decouples the VBW from the RBW and decreases it to smooth the trace.

//--------------Configuring the Sweep--------------------------
SENS:SWE:COUN 10
//Defines 10 sweeps to be performed in each measurement.
SENS:SWE:POIN 500
//During each sweep, 500 trace points will be measured.
SENS:SWE:TIME:AUTO OFF
SENS:SWE:TIME 50ms
//Decouples the sweep time from the RBW,VBW and span and increases it to
//make the measurement more precise.

//--------------Configuring Attenuation------------------------
//Only if electronic attenuator is available:
//INP:EATT:STAT ON
//Switches on the electronic attenuator.
//INP:EATT 30dB
//Sets the electronic attenuation to 30 dB.
INP:ATT 10dB
//Sets the mechanical attenuation to 10 dB and couples the reference level
//to the attenuation instead of vice versa.

//--------------Configuring the Amplitude and Scaling----------
DISP:TRAC1:Y:RLEV -10dBm
//Sets the reference level to -10 dBm.
DISP:TRAC1:Y:RLEV:OFFS 10dB
//Shifts the trace display in the diagram up by 10dB.
CALC:UNIT:POW V
//Sets the unit of the y-axis to Volt. The reference level is now 70.711 mV.
DISP:TRAC1:Y:SPAC LOG
//Uses logarithmic scaling with absolute values (V).
DISP:TRAC1:Y 110dB
//Increases the displayed range of the y-axis to 110 dB.
DISP:TRAC1:Y:RPOS 80PCT
//Shifts the display of the reference level down, it is no longer the top line
//in the diagram. The reference level is displayed as a red line.

//--------------Triggering-------------------------------------
```

```
TRIG:SOUR IFP
TRIG:LEV:IFP -10dBm
TRIG:SLOP POS
TRIG:DTIM 50ms
TRIG:IFP:HYST 5dB
TRIG:HOLD 10ms
//Defines triggering when the second intermediate frequency rises to a level
//of -10 dBm, with a dropout time of 50 ms, a hysteresis of 5 dB and a delay
//of 10 ms.


SWE:EGAT ON
SWE:EGAT:TYPE EDGE
SWE:EGAT:LENG 5ms
//Defines gating. Values are measured for 5 ms after triggering.

OUTP:TRIG2:DIR OUTP
OUTP:TRIG2:OTYP UDEF
OUTP:TRIG2:LEV HIGH
OUTP:TRIG2:PULS:LENG 100us
OUTP:TRIG2:PULS:IMM
//Configures a high trigger signal with a pulse length of 100 us to be output at
//the front TRIGGER INPUT/OUTPUT connector once.

//--------------Configuring the Trace--------------------------
DISP:TRAC2 ON
DISP:TRAC2:MODE AVER
DISP:TRAC3 ON
DISP:TRAC3:MODE MAXH
//Configures 3 traces: 1 (default): clear/write; 2: average; 3: max hold

SENS:DET1 POS
SENS:DET2 RMS
SENS:DET3 POS
//Configures traces 1 and 3 to use the positive peak detector; trace 2 uses
//the RMS detector.

TRAC:COPY TRACE4,TRACE1
//Copies trace 1 to a new trace 4 which will then be averaged.

SENS:AVER:STAT4 ON
SENS:AVER:COUN 10
SENS:AVER:TYPE LIN
//Configures trace 4 to be averaged linearly over 10 sweeps.

CALC:MATH:STAT ON
CALC:MATH:MODE LIN
CALC:MATH (TRACE1-TRACE2)
CALC:MATH:POS 100
//Calculates the linear difference between the measured and average values.
```

```
//The resulting trace is displayed at the top of the diagram.

//-------------Performing the Measurement--------------------
INIT;*WAI
//Initiates a new measurement and waits until the last sweep has finished.

//--------------Retrieving Results---------------------------
TRAC:DATA? TRACE1
TRAC:DATA? TRACE2
TRAC:DATA? TRACE3
TRAC:DATA? TRACE4
//Returns one power and one frequency value per sweep point for each trace.
```

10.3.2 Configuring Power Measurements

The following commands work for several power measurements.

CALCulate<n>:MARKer<m>:FUNCtion:POWer:MODE <Mode>

This command selects the trace display mode for power measurements.

Parameters:

<Mode>

WRITe
The power is calculated from the current trace.

MAXHold
The power is calculated from the current trace and compared with the previous power value using a maximum algorithm.

Manual control: See "Power Mode" on page 49

CALCulate<n>:MARKer<m>:FUNCtion:POWer:RESult? <Measurement>

This command queries the results of power measurements.

To get a valid result, you have to perform a complete measurement with synchronization to the end of the measurement before reading out the result. This is only possible for single sweeps. See also `INITiate:CONTinuous` on page 460.

Query parameters:

<Measurement>

ACPower | MCACpower
ACLR measurements (also known as adjacent channel power or multi-carrier adjacent channel measurements).
Returns the power for every active transmission and adajcent channel. The order is:
• power of the transmission channel
• power of lower adjacent channel
• power of upper adjacent channel
• power of lower alternate channel 1
• power of upper alternate channel 1 *(etc.)*
The unit of the return values depends on the scaling of the y-axis:
• logarithmic scaling returns the power in the current unit
• linear scaling returns the power in W

CN
Carrier-to-noise measurements.
Returns the C/N ratio in dB.

CN0
Carrier-to-noise measurements.
Returns the C/N ratio referenced to a 1 Hz bandwidth in dBm/Hz.

CPOWer
Channel power measurements.
Returns the channel power. The unit of the return values depends on the scaling of the y-axis:
• logarithmic scaling returns the power in the current unit
• linear scaling returns the power in W
For SEM measurements, the return value is the channel power of the reference range.

PPOWer
Peak power measurements.
Returns the peak power. The unit of the return values depends on the scaling of the y-axis:
• logarithmic scaling returns the power in the current unit
• linear scaling returns the power in W
For SEM measurements, the return value is the peak power of the reference range.

OBANdwidth | OBWidth
Occupied bandwidth.
Returns the occupied bandwidth in Hz.

Manual control: See "C/N" on page 66
See "C/No" on page 66

CALCulate<n>:MARKer<m>:FUNCtion:POWer:SELect <MeasType>

This command selects a power measurement and turns the measurement on.

Parameters:

<MeasType>	**ACPower | MCACpower** Adjacent channel leakage ratio (ACLR) aka adjacent channel power or multi carrier adjacent channel. The R&S FSW performs the measurement on the trace selected with [SENSe:]POWer:TRACe.
	CPOWer Channel power measurement with a single carrier. The R&S FSW performs the measurement on the trace selected with [SENSe:]POWer:TRACe.
	OBANdwidth | OBWidth Occupied bandwidth measurement. The R&S FSW performs the measurement on the trace that marker 1 is positioned on.
	CN Carrier-to-noise ratio measurement.
	CN0 Carrier-to-noise ratio measurement referenced to 1 Hz bandwidth

Manual control: See "C/N" on page 66
See "C/No" on page 66

CALCulate<n>:MARKer<m>:FUNCtion:POWer[:STATe] <State>

This command turns a power measurement on and off.

You can select a particular power measurement with CALCulate<n>:MARKer<m>:FUNCtion:POWer:SELect.

Parameters:

<State>	ON | OFF
	*RST: OFF

Manual control: See "C/N" on page 66
See "C/No" on page 66

[SENSe:]POWer:ACHannel:PRESet <Measurement>

This command determines the ideal span, bandwidths and detector for the current power measurement.

To get a valid result, you have to perform a complete measurement with synchronization to the end of the measurement before reading out the result. This is only possible for single sweeps. See also INITiate:CONTinuous on page 460.

Parameters:

<Measurement>
: **ACPower | MCACpower**
ACLR measurement

CPOWer
channel power measurement

OBANdwidth | OBWidth
Occupied bandwidth measurement

CN
Carrier to noise ratio

CN0
Carrier to noise ration referenced to a 1 Hz bandwidth

Manual control: See "Optimized Settings (Adjust Settings)" on page 49
See "Adjust Settings" on page 66
See "Adjust Settings" on page 71

[SENSe:]POWer:ACHannel:PRESet:RLEVel

This command determines the ideal reference level for the current measurement.

This automatic routine makes sure that the that the signal power level does not overload the R&S FSW or limit the dynamic range by too small a S/N ratio.

To determine the best reference level, the R&S FSW aborts current measurements and performs a series of test sweeps. After it has finished the test, it continues with the actual measurement.

To get a valid result, you have to perform a complete sweep with synchronization to the sweep end. This is only possible in single sweep mode.

Usage: Event

[SENSe:]POWer:TRACe <TraceNumber>

This command selects the trace channel power measurements are performed on.

For the measurement to work, the corresponding trace has to be active.

Parameters:

<TraceNumber>
: Range: 1 to 6
*RST: 1

Example: `POW:TRAC 2`
Assigns the measurement to trace 2.

Manual control: See "Selected Trace" on page 48

10.3.3 Measuring the Channel Power and ACLR

All remote control commands specific to channel power or ACLR measurements are described here.

10.3.3.1 Managing Measurement Configurations

The following commands control measurement configurations for ACLR measurements.

CALCulate<n>:MARKer<m>:FUNCtion:POWer:PRESet <Standard>

This command loads a measurement configuration.

The measurement configuration for power measurements consists of weighting filter, channel bandwidth and spacing, resolution and video bandwidth, detector and sweep time.

Parameters:

<Standard> For more information see chapter 4.2.7, "Reference: Predefined CP/ACLR Standards", on page 62.
If you want to load a customized configuration, the parameter is a string containing the file name.

Manual control: See "Standard" on page 45
See "Predefined Standards" on page 45
See "User-Defined Standards" on page 45

CALCulate<n>:MARKer<m>:FUNCtion:POWer:STANdard:CATalog?

This command queries all files containing ACLR standards.

Return values:

<Standards> List of standard files.

Usage: Query only

Manual control: See "Standard" on page 45
See "User-Defined Standards" on page 45

CALCulate<n>:MARKer<m>:FUNCtion:POWer:STANdard:DELete <Standard>

This command deletes a file containing an ACLR standard.

Parameters:

<Standard> String containing the file name of the standard.

Usage: Event

Manual control: See "Standard" on page 45
See "User-Defined Standards" on page 45

CALCulate<n>:MARKer<m>:FUNCtion:POWer:STANdard:SAVE <Standard>

This command saves the current ACLR measurement configuration as a new ACLR standard.

The measurement configuration for power measurements consists of weighting filter, channel bandwidth and spacing, resolution and video bandwidth, detector and sweep time.

Parameters:

<Standard> String containing the file name. The file format is xml.

Manual control: See "Standard" on page 45
See "User-Defined Standards" on page 45

10.3.3.2 Configuring the Channels

The following commands configure channels for channel power and ACLR measurements.

[SENSe:]POWer:ACHannel:ACPairs <ChannelPairs>

This command defines the number of pairs of adjacent and alternate channels.

Parameters:

<ChannelPairs> Range: 0 to 12
*RST: 1

Manual control: See "Number of Channels (TX, ADJ)" on page 46

[SENSe:]POWer:ACHannel:BANDwidth|BWIDth:ACHannel <Bandwidth>

This command defines the channel bandwidth of the adjacent channel.

The adjacent channel is the first pair of channels next to the transmission channels. If you set the channel bandwidth for these channels, the R&S FSW sets the bandwidth of the alternate channels to the same value.

Steep-edged channel filters are available for fast ACLR measurements.

Parameters:

<Bandwidth>	Range:	100 Hz to 1000 MHz
	*RST:	14 kHz

Manual control: See "Channel Bandwidths" on page 51

[SENSe:]POWer:ACHannel:BANDwidth|BWIDth:ALTernate<ch> <Bandwidth>

This command defines the channel bandwidth of the alternate channels.

If you set the channel bandwidth for the first alternate channel, the R&S FSW sets the bandwidth of the other alternate channels to the same value, but not the other way round. The command works hierarchically: to set a bandwidth of the 3rd and 4th channel, you have to set the bandwidth of the 3rd channel first.

Steep-edged channel filters are available for fast ACLR measurements.

Parameters:

<Bandwidth>	Range:	100 Hz to 1000 MHz
	*RST:	14 kHz

Manual control: See "Channel Bandwidths" on page 51

[SENSe:]POWer:ACHannel:BANDwidth|BWIDth[:CHANnel<ch>] <Bandwidth>

This command defines the channel bandwidth of the transmission channels.

Steep-edged channel filters are available for fast ACLR measurements.

Parameters:

<Bandwidth>	Range:	100 Hz to 1000 MHz
	*RST:	14 kHz

Manual control: See "Channel Bandwidths" on page 51
See "Channel Bandwidth" on page 66
See "Channel Bandwidth" on page 70

[SENSe:]POWer:ACHannel:NAME:ACHannel <Name>

This command defines a name for the adjacent channel.

Parameters:

<Name> String containing the name of the channel

*RST: ADJ

Manual control: See "Channel Names" on page 53

[SENSe:]POWer:ACHannel:NAME:ALTernate<ch> <Name>

This command defines a name for an alternate channel.

Parameters:

<Name> String containing the name of the channel
*RST: ALT<1...11>

Manual control: See "Channel Names" on page 53

[SENSe:]POWer:ACHannel:NAME:CHANnel<ch> <Name>

This command defines a name for a transmission channel.

Parameters:

<Name> String containing the name of the channel
*RST: TX<1...12>

Manual control: See "Channel Names" on page 53

[SENSe:]POWer:ACHannel:SPACing[:ACHannel] <Spacing>

This command defines the distance from transmission channel to adjacent channel.

A change of the adjacent channel spacing causes a change in the spacing of all alternate channels below the adjacent channel.

Parameters:

<Spacing> Range: 100 Hz to 2000 MHz
*RST: 14 kHz

Usage: SCPI confirmed

Manual control: See "Channel Spacings" on page 51

[SENSe:]POWer:ACHannel:SPACing:ALTernate<ch> <Spacing>

This command defines the distance from transmission channel to alternate channels.

If you set the channel spacing for the first alternate channel, the R&S FSW adjusts the spacing of alternate channels of a lower order, but not the other way round. The command works hierarchically: to set a distance from the transmission channel to the 2nd and 3rd alternate channel, you have to define a spacing for the 2nd alternate channel first.

Parameters:

<Spacing> Range: 100 Hz to 2000 MHz
*RST: 40 kHz (ALT1), 60 kHz (ALT2), 80 kHz (ALT3), ...

Manual control: See "Channel Spacings" on page 51

[SENSe:]POWer:ACHannel:SPACing:CHANnel<ch> <Spacing>

This command defines the distance between transmission channels.

If you set the channel spacing for a transmission channel, the R&S FSW sets the spacing of the lower transmission channels to the same value, but not the other way round. The command works hierarchically: to set a distance between the 2nd and 3rd and 3rd and 4th channel, you have to set the spacing between the 2nd and 3rd channel first.

Parameters:

<Spacing>	Range:	14 kHz to 2000 MHz
	*RST:	20 kHz

Manual control: See "Channel Spacings" on page 51

[SENSe:]POWer:ACHannel:TXCHannel:COUNt <Number>

This command defines the number of transmission channels.

The command works for measurements in the frequency domain.

Parameters:

<Number>	Range:	1 to 18
	*RST:	1

Manual control: See "Number of Channels (TX, ADJ)" on page 46

10.3.3.3 Defining Weighting Filters

The following commands define weighting filters for ACLR measurements.

[SENSe:]POWer:ACHannel:FILTer:ALPHa:ACHannel <Alpha>

This command defines the roll-off factor for the adjacent channel weighting filter.

Parameters:

<Alpha>	Roll-off factor	
	Range:	0 to 1
	*RST:	0.22

Manual control: See "Weighting Filters" on page 52

[SENSe:]POWer:ACHannel:FILTer:ALPHa[:ALL] <Value>

This command defines the alpha value for the weighting filter for all channels.

Parameters:

<Value> <numeric value>

*RST: 0,22

Example: `POW:ACH:FILT:ALPH:ALL 0,35`

[SENSe:]POWer:ACHannel:FILTer:ALPHa:ALTernate<ch> <Alpha>

This command defines the roll-off factor for the alternate channel weighting filter.

Parameters:

<Alpha> Roll-off factor

Range: 0 to 1
*RST: 0.22

Manual control: See "Weighting Filters" on page 52

[SENSe:]POWer:ACHannel:FILTer:ALPHa:CHANnel<ch> <Alpha>

This command defines the roll-off factor for the transmission channel weighting filter.

Parameters:

<Alpha> Roll-off factor

Range: 0 to 1
*RST: 0.22

Manual control: See "Weighting Filters" on page 52

[SENSe:]POWer:ACHannel:FILTer[:STATe]:ACHannel <State>

This command turns the weighting filter for the adjacent channel on and off.

Parameters:

<State> ON | OFF

*RST: OFF

Manual control: See "Weighting Filters" on page 52

[SENSe:]POWer:ACHannel:FILTer[:STATe]:ALL <State>

This command turns the weighting filters for all channels on and off.

Parameters:

<State> ON | OFF

*RST: OFF

[SENSe:]POWer:ACHannel:FILTer[:STATe]:ALTernate<ch> <State>

This command turns the weighting filter for an alternate channel on and off.

Parameters:

<State> ON | OFF

*RST: OFF

Manual control: See "Weighting Filters" on page 52

[SENSe:]POWer:ACHannel:FILTer[:STATe]:CHANnel<ch> <State>

This command turns the weighting filter for a transmission channel on and off.

Parameters:

<State> ON | OFF

*RST: OFF

Manual control: See "Weighting Filters" on page 52

10.3.3.4 Selecting the Reference Channel

The following commands define the reference channel for relative ACLR measurements.

[SENSe:]POWer:ACHannel:REFerence:AUTO ONCE

This command sets the channel power as the reference for relative ACLR measurements.

Example: `POW:ACH:REF:AUTO ONCE`

Usage: Event

Manual control: See "Setting a Fixed Reference for Channel Power Measurements (Set CP Reference)" on page 49

[SENSe:]POWer:ACHannel:REFerence:TXCHannel:AUTO <RefChannel>

This command selects the reference channel for relative measurements.

You need at least one channel for the command to work.

Parameters:

<RefChannel>	**MINimum** Transmission channel with the lowest power **MAXimum** Transmission channel with the highest power **LHIGhest** Lowest transmission channel for lower adjacent channels and highest transmission channel for upper adjacent channels

Example: `POW:ACH:REF:TXCH:AUTO MAX`
Selects the channel with the peak power as reference channel.

Manual control: See "Reference Channel" on page 47

[SENSe:]POWer:ACHannel:REFerence:TXCHannel:MANual <ChannelNumber>

This command defines a reference channel for relative ACLR measurements.

You need at least one channel for the command to work.

Parameters:

<ChannelNumber>	Range:	1 to 18
	*RST:	1

Manual control: See "Reference Channel" on page 47

10.3.3.5 Checking Limits

The following commands configure limit checks for channel power and ACLR measurements.

CALCulate<n>:LIMit<k>:ACPower:ACHannel:ABSolute <LowerLimit>, <UpperLimit>

This command defines the absolute limit of the adjacent channels.

If you have defined an absolute limit as well as a relative limit, the R&S FSW uses the lower value for the limit check.

Parameters:

<LowerLimit>, <UpperLimit>	The first value defines the limit of the lower adjacent channel, the second value the limit of the upper adjacent channel. Range: -200 dBm to 200 dBm *RST: -200 dBm

CALCulate<n>:LIMit<k>:ACPower:ACHannel:ABSolute:STATe <State>

This command turns the absolute limit check for the adjacent channels on and off.

You have to activate the general ACLR limit check before using this command with `CALCulate<n>:LIMit<k>:ACPower[:STATe]`.

Parameters:

<State>	ON | OFF *RST: OFF

CALCulate<n>:LIMit<k>:ACPower:ACHannel[:RELative] <LowerLimit>, <UpperLimit>

This command defines the relative limit of the adjacent channels. The reference value for the relative limit is the measured channel power.

If you have defined an absolute limit as well as a relative limit, the R&S FSW uses the lower value for the limit check.

Parameters:

<LowerLimit>, <UpperLimit>	The first value defines the limit of the lower adjacent channel, the second value the limit of the upper adjacent channel. Range: 0 dB to 100 dB *RST: 0 dB Default unit: dB

CALCulate<n>:LIMit<k>:ACPower:ACHannel:RESult?

This command queries the state of the limit check for the adjacent channels in an ACLR measurement.

To get a valid result, you have to perform a complete measurement with synchronization to the end of the measurement before reading out the result. This is only possible for single sweeps. See also `INITiate:CONTinuous` on page 460.

Return values:

<LowerACH>, <UpperACH>	text value The command returns two results. The first is the result for the lower, the second for the upper adjacent channel. **PASSED** Limit check has passed. **FAIL** Limit check has failed.

Example:
```
INIT:IMM;*WAI;
CALC:LIM:ACP:ACH:RES?
PASSED,PASSED
```

Usage: Query only

Manual control: See "Limit Checking" on page 52

CALCulate<n>:LIMit<k>:ACPower:ACHannel[:RELative]:STATe <State>

This command turns the relative limit check for the adjacent channels on and off.

You have to activate the general ACLR limit check before using this command with `CALCulate<n>:LIMit<k>:ACPower[:STATe]`.

Parameters:

<State> ON | OFF

*RST: OFF

CALCulate<n>:LIMit<k>:ACPower:ALTernate<ch>:ABSolute <LowerLimit>, <UpperLimit>

This command defines the absolute limit of the alternate channels.

If you have defined an absolute limit as well as a relative limit, the R&S FSW uses the lower value for the limit check.

Parameters:

<LowerLimit>, <UpperLimit> The first value defines the limit of the lower alternate channel, the second value the limit of the upper alternate channel.

Range: -200 dBm to 200 dBm
*RST: -200 dBm

CALCulate<n>:LIMit<k>:ACPower:ALTernate<ch>:ABSolute:STATe <State>

This command turns the absolute limit check for the alternate channels on and off.

You have to activate the general ACLR limit check before using this command with `CALCulate<n>:LIMit<k>:ACPower[:STATe]`.

Parameters:

<State> ON | OFF

*RST: OFF

CALCulate<n>:LIMit<k>:ACPower:ALTernate<ch>[:RELative] <LowerLimit>, <UpperLimit>

This command defines the relative limit of the alternate channels. The reference value for the relative limit is the measured channel power.

If you have defined an absolute limit as well as a relative limit, the R&S FSW uses the lower value for the limit check.

Parameters:

<LowerLimit>, <UpperLimit>
The first value defines the limit of the lower alternate channel, the second value the limit of the upper alternate channel.
Range: 0 dB to 100 dB
*RST: 0 DB
Default unit: dB

Manual control: See "Limit Checking" on page 52

CALCulate<n>:LIMit<k>:ACPower:ALTernate<ch>:RESult?

This command queries the state of the limit check for the adjacent or alternate channels in an ACLR measurement.

To get a valid result, you have to perform a complete measurement with synchronization to the end of the measurement before reading out the result. This is only possible for single sweeps. See also `INITiate:CONTinuous` on page 460.

Return values:

<LowerChan>, <UpperChan>
text value
The command returns two results. The first is the result for the lower, the second for the upper adjacent or alternate channel.
PASSED
Limit check has passed.
FAIL
Limit check has failed.

Example:
```
INIT:IMM;*WAI;
CALC:LIM:ACP:ACH:RES?
PASSED,PASSED
```

Usage: Query only

CALCulate<n>:LIMit<k>:ACPower:ALTernate<ch>[:RELative]:STATe <State>

This command turns the relative limit check for the alternate channels on and off.

You have to activate the general ACLR limit check before using this command with `CALCulate<n>:LIMit<k>:ACPower[:STATe]`.

Parameters:

<State>
ON | OFF
*RST: OFF

CALCulate<n>:LIMit<k>:ACPower[:STATe] <State>

This command turns the limit check for ACLR measurements on and off.

You have to use CALCulate<n>:LIMit<k>:ACPower:ACHannel[:RELative]:STATe or CALCulate<n>:LIMit<k>:ACPower:ALTernate<ch>[:RELative]:STATe in combination with this command to select the channels the limit check should be performed on.

Parameters:

<State>	ON | OFF	
	*RST:	OFF

Manual control: See "Limit Checking" on page 52

10.3.3.6 General ACLR Measurement Settings

The following commands control the measurement algorithm.

Useful commands for the ACLR measurement described elsewhere:

- [SENSe:]POWer:NCORrection on page 576

Remote commands exclusive to ACLR measurement

[SENSe:]POWer:HSPeed <State>

This command turns high speed ACLR and channel power measurements on and off.

If on, the R&S FSW performs a measurement on each channel in the time domain. It returns to the frequency domain when the measurement is done.

In some telecommunications standards, high speed measurements use weighting filters with characteristic or steep-edged channel filters for band limitation.

Parameters:

<State>	ON | OFF	
	*RST:	OFF

Example: `POW:HSP ON`

Manual control: See "Fast ACLR" on page 48

10.3.3.7 Analyzing Measurement Results

The following commands analyze and retrieve measurement results for ACLR measurements.

Useful commands for channel power measurements described elsewhere

- CALCulate<n>:MARKer<m>:FUNCtion:POWer:RESult? on page 464
- TRACe<n>[:DATA] on page 625

Remote commands exclusive to channel power measurements

CALCulate<n>:MARKer<m>:FUNCtion:POWer:RESult:PHZ <State>

This command selects the way the R&S FSW returns results for power measurements.

You can query results with `CALCulate<n>:MARKer<m>:FUNCtion:POWer:RESult?`.

Parameters:
<State> ON | OFF

ON
Channel power density in dBm/Hz

OFF
Channel power in dBm

*RST: OFF

Example: `CALC:MARK:FUNC:POW:RES:PHZ ON`
Output of results referred to the channel bandwidth.

Manual control: See "Channel Power Levels and Density (Power Unit)" on page 48

[SENSe:]POWer:ACHannel:MODE <Mode>

This command selects the way the R&S FSW displays the power of adjacent channels.

You need at least one adjacent channel for the command to work.

Parameters:
<Mode> **ABSolute**
Shows the absolute power of all channels

RELative
Shows the power of adjacent and alternate channels in relation to the transmission channel

*RST: RELative

Manual control: See "Absolute and Relative Values (ACLR Mode)" on page 48

10.3.3.8 Example: Configuring and Performing Channel Power Measurements

In this example we will configure and perform an adjacent-channel power measurement. Note that this example is primarily meant to demonstrate the remote control commands, it does not necessarily reflect a useful measurement task. For most common measurement standards, the R&S FSW performs the measurement optimally with the predefined settings, without further configuration.

```
//-------------Preparing the measurement---------------------
*RST
```

```
//Resets the instrument
INIT:CONT OFF
//Selects single sweep mode.

CALC:MARK:FUNC:POW:SEL ACP
//Activates adjacent-channel power measurement.
CALC:MARK:FUNC:POW:PRES GSM
//Selects the user standard "GSM"

//---------------Setting Up Channels------------
POW:ACH:TXCH:COUN 1
//Creates one transmission channel.
POW:ACH:NAME:CHAN1 'TX Channel'
//Names the first transmission channel 'TX Channel'.
POW:ACH:ACP 2
//Creates two adjacent channels - one adjacent channel and one alternate channel.
POW:ACH:NAME:ACH 'ABC'
//Names the adjacent channel 'ABC'
POW:ACH:NAME:ALT1 'XYZ'
//Names the first alternate channel 'XYZ'.
POW:ACH:BWID:CHAN1 30kHz
//Defines a bandwidth of 30 kHz for the transmission channel.
POW:ACH:BWID:ACH 30kHz
//Defines a bandwidth of 30 kHz for the adjacent channel.
POW:ACH:BWID:ALT1 30kHz
//Defines a bandwidth of 30 kHz for the first alternate channel.
POW:ACH:SPAC 33kHz
//Defines a distance of 33 kHz from the center of the transmission channel to the
//center of the adjacent channel.
//Also adjusts the distance to the alternate channels (66 kHz).
POW:ACH:SPAC:ALT1 100kHz
//Defines a distance of 100 kHz from the center of the transmission channel to the
//center of the first alternate channel.

//---------------Selecting a Reference Channel--
POW:ACH:MODE REL
//Selects relative display of the channel power.
POW:ACH:REF:TXCH:MAN 1
//Defines transmission channel 1 as the reference channel.

//-----------Saving the settings as a user standard--------------

CALC:MARK:FUNC:POW:STAN:SAVE 'my_aclr_standard'
//Saves the user standard with the name "my_aclr_standard".
//Weighting filters can only be defined for user-defined standards.

//---------------Defining Weighting Filters-----
POW:ACH:FILT:ALPH:CHAN1 0.35
//Defines a roll-off factor of 0.35 for the weighting filter of the first
//transmission channel.
```

```
POW:ACH:FILT:CHAN1 ON
//Turns the weighting filter for the first transmission channel on.
POW:ACH:FILT:ALPH:ACH 0.35
//Defines a roll-off factor of 0.35 for the weighting filter of the adjacent
//channel.
POW:ACH:FILT:ACH ON
//Turns the weighting filter for the adjacent channel on.
POW:ACH:FILT:ALPH:ALT1 0.35
//Defines a roll-off factor of 0.35 for the weighting filter of the first
//alternate channel.
POW:ACH:FILT:ALT1 ON
//Turns the weighting filter for the first alternate channel on.

//---------------Working with Limits------------
CALC:LIM:ACP:ACH 30DB,30DB
//Defines a relative limit of 30 dB below the power of the reference channel
//for both adjacent channels.
CALC:LIM:ACP:ALT1 25DB,25DB
//Defines a relative limit of 25 dB below the power of the reference channel
//for the first alternate channels.
CALC:LIM:ACP:ACH:ABS -35DBM,-35DBM
//Defines an absolute limit of -35 dBm for both adjacent channels.
CALC:LIM:ACP ON
//Turns the ACLR limit check on.
CALC:LIM:ACP:ACH:STAT ON
//Turns the relative limit check for adjacent channels on.
CALC:LIM:ACP:ACH:ABS:STAT ON
//Turns the absolute limit check for adjacent channels on.
CALC:LIM:ACP:ALT1:ABS:STAT ON
//Turns the absolute limit check for the first alternate channel on.

//--------------Performing the Measurement-----
POW:ACH:PRES ACP;*WAI
//Determines the ideal ACLR measurement configuration.
POW:ACH:PRES:RLEV;*WAI
//Determines the ideal reference level for the measurement.
INIT;*WAI
//Initiates a new measurement and waits until the sweep has finished.

//---------------Limit Check--------------------
CALC:LIM:ACP:ACH:RES?
//Queries the results of the limit check for the adjacent channels.
CALC:LIM:ACP:ALT1:RES?
//Queries the results of the limit check for the first alternate channels.

//---------------Retrieving Results-------------
CALC:MARK:FUNC:POW:RES? ACP
//Returns the results for the ACLR measurement.
```

10.3.4 Measuring the Carrier-to-Noise Ratio

The following commands are necessary to perform carrier-to-noise measurements.

- CALCulate<n>:MARKer<m>:FUNCtion:POWer[:STATe]
- CALCulate<n>:MARKer<m>:FUNCtion:POWer:RESult?
- CALCulate<n>:MARKer<m>:FUNCtion:POWer:SELect
- [SENSe:]POWer:ACHannel:BANDwidth|BWIDth[:CHANnel<ch>]
- [SENSe:]POWer:ACHannel:PRESet

Programming example

This programming example demonstrates how to perform a Carrier-to-noise measurement in a remote environment.

```
//----------Preparing the measurement------------
*RST
//Reset the instrument
FREQ:CENT 800MHz
//Sets the center frequency to the carrier frequency of 800 MHz.
CALC:MARK:FUNC:POW:SEL CN
//Activates carrier-to-noise ratio measurement.
POW:ACH:PRES CN
//Optimizes the instrument settings according to the channel bandwidth.
POW:ACH:PRES:RLEV
//Determines the ideal reference level for the measurement.

//--------------Performing the Measurement-----
INIT:CONT OFF
//Selects single sweep mode.
INIT;*WAI
//Initiates a new measurement and waits until the sweep has finished.

// Now turn off the carrier signal and repeat the measurement:
INIT;*WAI
//Initiates a new measurement and waits until the sweep has finished.

//---------------Retrieving Results-------------
CALC:MARK:FUNC:POW:RES? CN
//Returns the carrier-to-noise ratio.
```

10.3.5 Measuring the Occupied Bandwidth

All remote control commands specific to occupied bandwidth measurements are described here.

10.3.5.1 Configuring the Measurement

The following commands configure measurements of the occupied bandwidth.

Useful commands for occupied bandwidth measurements described elsewhere

Configuring the channel:

- `[SENSe:]POWer:ACHannel:BANDwidth|BWIDth[:CHANnel<ch>]`
- `[SENSe:]POWer:ACHannel:PRESet`
- `[SENSe:]POWer:ACHannel:PRESet:RLEVel`

Defining search limits:

- `CALCulate:MARKer:X:SLIMits[:STATe]` on page 634
- `CALCulate:MARKer:X:SLIMits:LEFT` on page 635
- `CALCulate:MARKer:X:SLIMits:RIGHT` on page 635

Retrieving results:

- `CALCulate<n>:MARKer<m>:FUNCtion:POWer:RESult?` on page 464
- `CALCulate<n>:MARKer<m>:FUNCtion:POWer:SELect` on page 465
- `CALCulate<n>:MARKer<m>:FUNCtion:POWer[:STATe]` on page 466

Remote commands exclusive to occupied bandwidth measurements

[SENSe:]POWer:BANDwidth|BWIDth <Percentage>

This command selects the percentage of the total power that defines the occupied bandwidth.

Parameters:

<Percentage>	Range:	10 PCT to 99.9 PCT
	*RST:	99 PCT

Example: `POW:BWID 95PCT`

Manual control: See "% Power Bandwidth" on page 70

10.3.5.2 Programming Example: OBW Measurement

This programming example demonstrates the measurement example described in chapter 4.4.5, "Measurement Example", on page 72 in a remote environment.

```
//-------------Configuring the Measurement-----------------------
*RST
//Resets the instrument
FREQ:CENT 800MHz
//Sets the center frequency to 800 MHz.
DISP:TRAC:Y:RLEV -10dBm
//Sets the reference level to -10 dBm.
```

```
CALC:MARK:FUNC:POW:SEL OBW
//Activates occupied bandwidth measurement.
POW:BWID 99PCT
//Sets the percentage of power to 99%.
POW:ACH:BAND 21kHz
//Sets the channel bandwidth to 21 kHz.
POW:ACH:PRES OBW
//Optimizes the instrument settings according to the channel bandwidth.
POW:ACH:PRES:RLEV
//Determines the ideal reference level for the measurement.
DET APE
//Sets the trace detector to positive peak.

//--------------Performing the Measurement-----
INIT:CONT OFF
//Selects single sweep mode.
INIT;*WAI
//Initiates a new measurement and waits until the sweep has finished.

//---------------Retrieving Results-------------
CALC:MARK:FUNC:POW:RES? OBW
//Returns the occupied bandwidth.
```

10.3.6 Measuring the Spectrum Emission Mask

All remote control commands specific to spectrum emission mask measurements are described here.

10.3.6.1 Managing Measurement Configurations

The following commands control measurement configurations for SEM measurements.

CALCulate<n>:LIMit:ESPectrum:RESTore

This command restores the predefined limit lines for the selected Spectrum Emission Mask standard.

All modifications made to the predefined limit lines are lost and the factory-set values are restored.

Example: `CALC:LIM:ESP:REST`
Resets the limit lines for the current Spectrum Emission Mask standard to the default setting.

[SENSe:]ESPectrum:PRESet[:STANdard] <Standard>

This command loads a measurement configuration.

Standard definitions are stored in an xml file. The default directory for SEM standards is `C:\r_s\instr\sem_std`.

Parameters:
<Standard> String containing the file name.
If you have stored the file in a subdirectory of the directory mentioned above, you have to include the relative path to the file.

Return values:
<Standard> The query returns the name of the currently loaded standard.

[SENSe:]ESPectrum:PRESet:RESTore

This command restores the default configurations of predefined SEM standards.

Note that the command will overwrite customized standards that have the same name as predefined standards.

Usage: Event

Manual control: See "Restore Standard Files" on page 92

[SENSe:]ESPectrum:PRESet:STORe <Standard>

This command saves the current SEM measurement configuration.

Standard definitions are stored in an xml file. The default directory for SEM standards is `C:\r_s\instr\sem_std`.

Parameters:
<Standard> String containing the file name.
You can save the file in a subdirectory of the directory mentioned above. In that case, you have to include the relative path to the file.

10.3.6.2 Controlling the Measurement

The following commands control the measurement itself.

INITiate:ESPectrum

This command initiates a Spectrum Emission Mask measurement.

Usage: Event

[SENSe:]SWEep:MODE <Mode>

This command selects the spurious emission and spectrum emission mask measurements.

You can select other measurements with

- CALCulate<n>:MARKer<m>:FUNCtion:POWer[:STATe]

Parameters:

<Mode>

AUTO
Turns on basic spectrum measurements.

ESPectrum
Turns on spectrum emission mask measurements.

LIST
Turns on spurious emission measurements.

*RST: AUTO

Usage: SCPI confirmed

Manual control: See "Spectrum Emission Mask" on page 32
See "Spurious Emissions" on page 32

10.3.6.3 Configuring a Sweep List

The following commands define a sweep list for SEM measurements.

See also:

- CALCulate:LIMit:ESPectrum:PCLass<class>:LIMit[:STATe] on page 503

[SENSe:]ESPectrum:HighSPeed <State>

This command turns high speed mode for SEM measurements on and off.

For more information including restrictions see chapter 4.5.4.3, "Fast SEM Measurements", on page 80.

Parameters:

<State> ON | OFF
*RST: OFF

Example: `ESP:HSP ON`

Manual control: See "Fast SEM" on page 83

[SENSe:]ESPectrum:RANGe<range>:BANDwidth[:RESolution] <RBW>

This command defines the resolution bandwidth for a SEM range.

In case of high speed measurements, the resolution bandwidth has to be identical for all ranges.

Suffix:

<range> 1...20
Selects the measurement range.

Parameters:

<RBW> Resolution bandwidth.
Refer to the data sheet for available resolution bandwidths.
*RST: 30.0 kHz
Default unit: Hz

Manual control: See "RBW" on page 83

[SENSe:]ESPectrum:RANGe<range>:BANDwidth:VIDeo <VBW>

This command defines the video bandwidth for a SEM range.

In case of high speed measurements, the video bandwidth has to be identical for all ranges.

Suffix:

<range>	1...20 Selects the measurement range.

Parameters:

<VBW>	Video bandwidth. Refer to the data sheet for available video bandwidths. *RST: 10.0 MHz Default unit: Hz

Manual control: See "VBW" on page 84

[SENSe:]ESPectrum:RANGe:COUNt? <Ranges>

This command queries the number of ranges in the sweep list.

Return values:

<Ranges>	Number of ranges in the sweep list.

Usage: Query only

[SENSe:]ESPectrum:RANGe<range>:DELete

This command removes a range from the sweep list.

Note that

- you cannot delete the reference range
- a minimum of three ranges is mandatory.

Suffix:

<range>	1...20 Selects the measurement range.

Usage: Event

Manual control: See "Delete Range" on page 86

[SENSe:]ESPectrum:RANGe<range>:FILTer:TYPE <FilterType>

This command selects the filter type for a SEM range.

In case of high speed measurements, the filter has to be identical for all ranges.

Suffix:

<range>	1...20 Selects the measurement range.

Parameters:

<FilterType>

NORMal
Gaussian filters

CFILter
channel filters

RRC
RRC filters

P5
5 Pole filters

*RST: NORM

Refer to the datasheet for available filter bandwidths.

Manual control: See "Filter Type" on page 83

[SENSe:]ESPectrum:RANGe<range>[:FREQuency]:STARt <Frequency>

This command defines the start frequency of a SEM range.

Make sure to set an appropriate span. If you set a span that is

- smaller than the span the SEM sweep list covers, the R&S FSW will not measure the ranges that are outside the span - results may be invalid.
- greater than the span the SEM sweep list covers, the R&S FSW will adjust the start frequency of the first SEM range and the stop frequency of the last SEM range to the span

For more information see chapter 4.5.4.1, "Ranges and Range Settings", on page 76.

Suffix:

<range> 1...20
Selects the measurement range.

Parameters:

<Frequency> Numeric value. Note that the minimum frequency range of a SEM range is 20 Hz.

*RST: -12.75 MHz (range 1), -2.515 MHz (range 2), 2.515 MHz (range 3)

Default unit: Hz

Manual control: See "Range Start / Range Stop" on page 83

[SENSe:]ESPectrum:RANGe<range>[:FREQuency]:STOP <Frequency>

This command defines the stop frequency of a SEM range.

Make sure to set an appropriate span. If you set a span that is

- smaller than the span the SEM sweep list covers, the R&S FSW will not measure the ranges that are outside the span - results may be invalid.
- greater than the span the SEM sweep list covers, the R&S FSW will adjust the start frequency of the first SEM range and the stop frequency of the last SEM range to the span

For more information see chapter 4.5.4.1, "Ranges and Range Settings", on page 76.

Suffix:
<range> 1...20
Selects the measurement range.

Parameters:
<Frequency> Numeric value.
*RST: -2.52 MHz (range 1), 2.52 MHz (range 2), 250.0 MHz (range 3)
Default unit: Hz

Manual control: See "Range Start / Range Stop" on page 83

[SENSe:]ESPectrum:RANGe<range>:INPut:ATTenuation <Attenuation>

This command defines the input attenuation for a SEM range.

In case of high speed measurements, the input attenuation has to be identical for all ranges.

Suffix:
<range> 1...20
Selects the measurement range.

Parameters:
<Attenuation> Numeric value.
Refer to the data sheet for the attenuation range.
*RST: 10 dB
Default unit: dB

Manual control: See "RF Attenuator" on page 84

[SENSe:]ESPectrum:RANGe<range>:INPut:ATTenuation:AUTO <State>

This command turns automatic selection of the input attenuation for a SEM range on and off.

In case of high speed measurements, the input attenuation has to be identical for all ranges.

Suffix:
<range> 1...20
Selects the measurement range.

Parameters:
<State> ON | OFF
*RST: ON

Example: `ESP:RANG2:INP:ATT:AUTO OFF`
Deactivates the RF attenuation auto mode for range 2.

Manual control: See "RF Att. Mode" on page 84

[SENSe:]ESPectrum:RANGe<range>:INPut:GAIN <Gain>

This command selects the level of preamplification for a SEM range.

In case of high speed measurements, the level of the preamplifier has to be identical for all ranges.

Suffix:

<range>	1...20 Selects the measurement range.

Parameters:

<Gain>	15 dB | 30 dB The availability of preamplification levels depends on the R&S FSW model. • R&S FSW8: 15dB and 30 dB • R&S FSW13: 15dB and 30 dB • R&S FSW26: 30 dB All other values are rounded to the nearest of these two. *RST: OFF

[SENSe:]ESPectrum:RANGe<range>:INPut:GAIN:STATe <State>

This command turns the preamplifier for a SEM range on and off.

In case of high speed measurements, the state of the preamplifier has to be identical for all ranges.

Suffix:

<range>	1...20 Selects the measurement range.

Parameters:

<State>	ON | OFF *RST: OFF
Manual control:	See "Preamp" on page 84

[SENSe:]ESPectrum:RANGe<range>:INSert <Mode>

This command inserts a new SEM range and updates the range numbers accordingly.

Suffix:

<range>	1...20 Selects the SEM range.

Parameters:

<Mode>	**AFTer** Inserts a range after the selected range. **BEFore** Inserts a range before the selected range.
Manual control:	See "Insert before/after Range" on page 86

[SENSe:]ESPectrum:RANGe<range>:LIMit<PClass>:ABSolute:STARt <Level>

This command defines an absolute limit for a SEM range.

Unlike manual operation, you can define an absolute limit anytime and regardless of the limit check mode.

Suffix:

<range> 1...20
Selects the measurement range.

<PClass> 1..4
Power class for which the limit is defined.

Parameters:

<Level> Absolute limit at the start frequency of a SEM range.
Range: -400 to 400
*RST: -13
Default unit: dBm

Example: `SENSe:ESPectrum:RANGe:LIMit:ABSolute:STARt -10`
For a detailed example see chapter 10.3.6.9, "Example: SEM Measurement", on page 508.

Manual control: See "Abs Limit Start/Stop" on page 85

[SENSe:]ESPectrum:RANGe<range>:LIMit<PClass>:ABSolute:STOP <Level>

This command defines an absolute limit for a SEM range.

Unlike manual operation, you can define an absolute limit anytime and regardless of the limit check mode.

Suffix:

<range> 1...20
Selects the measurement range.

<PClass> 1..4
Power class for which the limit is defined.

Parameters:

<Level> Absolute limit at the stop frequency of a SEM range.
Range: -400 to 400
*RST: -13
Default unit: dBm

Example: `SENSe:ESPectrum:RANGe:LIMit:ABSolute:STOP -15`
For a detailed example see chapter 10.3.6.9, "Example: SEM Measurement", on page 508.

Manual control: See "Abs Limit Start/Stop" on page 85

[SENSe:]ESPectrum:RANGe<range>:LIMit<PClass>:RELative:STARt <Limit>

This command defines a relative limit for a SEM range.

Unlike manual operation, you can define a relative limit regardless of the limit check mode.

Suffix:

<range>	1...20 Selects the SEM range.
<PClass>	1..4 Power class for which the limit is defined.

Parameters:

<Level>	Relative limit at the start frequency of a SEM range. Range: -400 to 400 *RST: -50 Default unit: dBc
Example:	`SENSe:ESPectrum:RANGe:LIMit:RELative:STARt -10` For a detailed example see chapter 10.3.6.9, "Example: SEM Measurement", on page 508.
Manual control:	See "Rel Limit Start/Stop" on page 85

[SENSe:]ESPectrum:RANGe<range>:LIMit<PClass>:RELative:STARt:ABS <Limit>

This command defines an absolute limit for the MAX function of the relative limit for a SEM range.

For more information see "Relative limit line functions" on page 79.

Suffix:

<range>	1...20 Selects the SEM range.
<PClass>	1..4 Power class for which the limit is defined.

Parameters:

<Level>	Absolute limit at the start frequency of a SEM range to be used in addition to the relative limit if the MAX function is enabled (see `[SENSe:]ESPectrum:RANGe<range>:LIMit<PClass>:RELative:STARt:FUNCtion` on page 495). Range: -400 to 400 *RST: -13 Default unit: dBm
Example:	`SENSe:ESPectrum:RANGe:LIMit:RELative:STARt:ABSolute -10` For a detailed example see chapter 10.3.6.9, "Example: SEM Measurement", on page 508.
Manual control:	See "Rel Limit Start/Stop" on page 85

[SENSe:]ESPectrum:RANGe<range>:LIMit<PClass>:RELative:STARt:FUNCtion <Function>

This command enables the use of a function when defining the relative limit for a SEM range.

Suffix:

<range>	1...20 Selects the SEM range.
<PClass>	1..4 Power class for which the limit is defined.

Parameters:

<Function>: Defines the function to be used to determine the relative limit line start value

MAX
The maximum of the relative *and* the absolute level is used as the limit start value. Use the `[SENSe:]ESPectrum:RANGe<range>:LIMit<PClass>:RELative:STARt` and `[SENSe:]ESPectrum:RANGe<range>:LIMit<PClass>:RELative:STARt:ABS` commands to define these values.

OFF
No function is used, the relative limit line is defined by a fixed relative start value. Use the `[SENSe:]ESPectrum:RANGe<range>:LIMit<PClass>:RELative:STARt` command to define this value.

*RST: OFF

Example:
`SENSe:ESPectrum:RANGe:LIMit:RELative:STARt:FUNCtion MAX`
For a detailed example see chapter 10.3.6.9, "Example: SEM Measurement", on page 508.

Manual control: See "Rel Limit Start/Stop" on page 85

[SENSe:]ESPectrum:RANGe<range>:LIMit<PClass>:RELative:STOP <Limit>

This command defines a relative limit for a SEM range.

Unlike manual operation, you can define a relative limit anytime and regardless of the limit check mode.

Suffix:

<range>	1...20 Selects the SEM range.
<PClass>	1..4 Power class for which the limit is defined.

Parameters:

<Level>	Relative limit at the stop frequency of a SEM range. Range: -400 to 400 *RST: -50 Default unit: dBc
Example:	`SENSe:ESPectrum:RANGe:LIMit:RELative:STOP -15` For a detailed example see chapter 10.3.6.9, "Example: SEM Measurement", on page 508.
Manual control:	See "Rel Limit Start/Stop" on page 85

[SENSe:]ESPectrum:RANGe<range>:LIMit<PClass>:RELative:STOP:ABSolute <Limit>

This command defines an absolute limit for the MAX function of the relative limit for a SEM range.

For more information see "Relative limit line functions" on page 79.

Suffix:

<range>	1...20 Selects the SEM range.
<PClass>	1..4 Power class for which the limit is defined.

Parameters:

<Level>	Absolute limit at the stop frequency of a SEM range to be used in addition to the relative limit if the MAX function is enabled (see `[SENSe:]ESPectrum:RANGe<range>:LIMit<PClass>:RELative:STOP:FUNCtion` on page 496). Range: -400 to 400 *RST: -13 Default unit: dBm
Example:	`SENSe:ESPectrum:RANGe:LIMit:RELative:STOP:ABSolute -15` For a detailed example see chapter 10.3.6.9, "Example: SEM Measurement", on page 508.
Manual control:	See "Rel Limit Start/Stop" on page 85

[SENSe:]ESPectrum:RANGe<range>:LIMit<PClass>:RELative:STOP:FUNCtion <Function>

This command enables the use of a function when defining the relative limit for a SEM range.

Suffix:

<range>	1...20 Selects the SEM range.

<PClass> 1..4
Power class for which the limit is defined.

Parameters:

<Function> Defines the function to be used to determine the relative limit line stop value

MAX
The maximum of the relative *and* the absolute level is used as the limit stop value. Use the `[SENSe:]ESPectrum:RANGe<range>:LIMit<PClass>:RELative:STOP` and `[SENSe:]ESPectrum:RANGe<range>:LIMit<PClass>:RELative:STOP:ABSolute` commands to define these values.

OFF
No function is used, the relative limit line is defined by a fixed relative stop value. Use the `[SENSe:]ESPectrum:RANGe<range>:LIMit<PClass>:RELative:STOP` command to define this value.

*RST: OFF

Example: `SENSe:ESPectrum:RANGe:LIMit:RELative:STOP:FUNCtion MAX`

For a detailed example see chapter 10.3.6.9, "Example: SEM Measurement", on page 508.

Manual control: See "Rel Limit Start/Stop" on page 85

[SENSe:]ESPectrum:RANGe:LIMit<PClass>:STATe <State>

This command selects the limit check mode for all SEM ranges.

Suffix:

<PClass> 1..4
Power class for which the limit is defined.

Parameters:

<State> ABSolute | RELative | AND | OR

ABSolute
Checks only the absolute limits defined.

RELative
Checks only the relative limits. Relative limits are defined as relative to the measured power in the reference range.

AND
Combines the absolute and relative limit. The limit check fails when both limits are violated.

OR
Combines the absolute and relative limit. The limit check fails when one of the limits is violated.

*RST: RELative

Manual control: See "Limit Check 1-4" on page 85

[SENSe:]ESPectrum:RANGe<range>:RLEVel <RefLevel>

This command defines the reference level for a SEM range.

In case of high speed measurements, the reference level has to be identical for all ranges.

Suffix:

<range>	1...20 Selects the measurement range.

Parameters:

<RefLevel>	Reference level. Refer to the data sheet for the reference level range. *RST: 0 dBm

Manual control: See "Ref. Level" on page 84

[SENSe:]ESPectrum:RANGe<range>:SWEep:TIME <SweepTime>

This command defines the sweep time for a SEM range.

In case of high speed measurements, the sweep time has to be identical for all ranges.

Suffix:

<range>	1...20 Selects the measurement range.

Parameters:

<SweepTime>	Sweep time. The range depends on the ratios of the span to the RBW and the RBW to the VBW. Refer to the data sheet for more information.

Manual control: See "Sweep Time" on page 84

[SENSe:]ESPectrum:RANGe<range>:SWEep:TIME:AUTO <State>

This command turns automatic selection of the sweep time for a SEM range on and off.

In case of high speed measurements, the sweep time has to be identical for all ranges.

Suffix:

<range>	1...20 Selects the measurement range.

Parameters:

<State>	ON | OFF *RST: ON

Example: `ESP:RANG3:SWE:TIME:AUTO OFF`
Deactivates the sweep time auto mode for range 3.

Manual control: See "Sweep Time Mode" on page 84

[SENSe:]ESPectrum:RANGe<range>:TRANsducer <Transducer>

This command selects a transducer factor for a SEM range.

Note that

- the transducer must cover at least the span of the range
- the x-axis has to be linear
- the unit has to be dB

Suffix:

<range>	1...20 Selects the measurement range.

Parameters:

<Transducer>	String containing the transducer file name, including the path information.

Manual control: See "Transd. Factor" on page 85

10.3.6.4 Configuring the Reference Range

The following commands define the reference range for the SEM sweep list.

[SENSe:]ESPectrum:BWID <Bandwidth>

This command defines the channel bandwidth of the reference range.

The bandwidth is available if the power reference is the channel power.

Parameters:

<Bandwidth>	minimum span ≤ value ≤ span of reference range *RST: 3.84 MHz

Manual control: See "Channel Power Settings" on page 88
See "Tx Bandwidth" on page 88

[SENSe:]ESPectrum:FILTer[:RRC]:ALPHa <Alpha>

This command defines the roll-off factor for the RRC filter.

The RRC filter is available if the power reference is the channel power.

Parameters:

<Alpha>	Range: 0 to 1 *RST: 0.22

Manual control: See "Channel Power Settings" on page 88
See "Alpha" on page 88

[SENSe:]ESPectrum:FILTer[:RRC][:STATe] <State>

This command turns the RRC filter in the reference range on and off.

The RRC filter is available if the power reference is the channel power.

Parameters:

<State> ON | OFF

*RST: ON

Manual control: See "Channel Power Settings" on page 88
See "RRC Filter State" on page 88

[SENSe:]ESPectrum:RRANge? <RefRange>

This command queries the reference range.

Return values:

<RefRange> Number of the current reference range.

Range: 1 to 20

Usage: Query only

[SENSe:]ESPectrum:RTYPe <Type>

This command defines the type of the power reference.

Parameters:

<Type> **PEAK**
Measures the highest peak within the reference range.

CPOWer
Measures the channel power within the reference range (integral bandwidth method).

*RST: CPOWer

Manual control: See "Power Reference Type" on page 87

10.3.6.5 Configuring the Power Classes

The following commands define the power classes for SEM measurements.

CALCulate:LIMit:ESPectrum:LIMits <Max1>,<Max2>,<Max3>

This command sets or queries up to 4 power classes in one step. You can only define values for the number of power classes defined by `CALCulate:LIMit:ESPectrum:PCLass:COUNt` on page 502.

Setting parameters:

<Max1> Defines the value range for power class 1 as `-200 to <Max1>`.
Only available for `CALC:LIM:ESP:PCL:COUNT >=2`
If only 2 power classes are defined, the value range for power class 2 is defined as `<Max1> to 200`.

Range: -199 to + 199

<Max2> Defines the value range for power class 2 as `<Max1> to <Max2>`.
Only available for `CALC:LIM:ESP:PCL:COUNT >=3`
If only 3 power classes are defined, the value range for power class 3 is defined as `<Max2> to 200`.

Range: -199 to + 199, <Max2> must be higher than <Max1>

<Max3> Defines the value range for power class 3 as `<Max2> to <Max3>`.
The value range for power class 4 is defined as `<Max3> to 200`.
Only available for `CALC:LIM:ESP:PCL:COUNT = 4`

Range: -199 to + 199, <Max3> must be higher than <Max2>

Return values:

<RangeLimits> As a result of the query, the range limits including the beginning of the first power class (-200) and the end of the last power class (+200) are indicated.

Example:

```
CALC:LIM:ESP:LIM -50,50,70
```

Defines the following power classes:

```
<-200, -50>
<-50, 50>
<50, 70>
<70, 200>
```

Query:

```
CALC:LIM:ESP:LIM?
```

Response:

```
-200,-50,50,70,200
```

Mode: A, CDMA, EVDO, TDS, WCDMA

CALCulate:LIMit:ESPectrum:MODE <Mode>

Which limit line is to be used for an SEM measurement depends on the power class the input signal power belongs to. This command defines wether the power class is determined automatically or manually.

Parameters:

<Mode>

AUTO
The power class (and thus the limit line) is assigned dynamically according to the currently measured channel power.

MANUAL
One of the specified power classes is selected manually for the entire measurement. The selection is made with the `CALCulate:LIMit:ESPectrum:PCLass<class>[:EXCLusive]` command.

*RST: AUTO

Example: `CALC:LIM:ESP:MODE AUTO`
Activates automatic selection of the limit line.

CALCulate:LIMit:ESPectrum:VALue <Power>

This command activates the manual limit line selection as and specifies the expected power as a value. Depending on the entered value, the associated predefined limit lines is selected.

This command has the same effect as a combination of the CALC:LIM:ESP:MODE MAN and the `CALCulate:LIMit:ESPectrum:PCLass<class>[:EXCLusive]` commands; however, the power class to be used is not defined directly, but via the expected power. As opposed to `CALC:LIM:ESP:MODE AUTO`, the power class is not reassigned to the input signal power dynamically, but only once when the command is executed.

Parameters:

<Power> integer

Range: -200 to 199
*RST: 0

Example: `CALC:LIM:ESP:VAL 33`
Activates manual selection of the limit line and selects the limit line for P = 33.

CALCulate:LIMit:ESPectrum:PCLass:COUNt <NoPowerClasses>

This command sets the number of power classes to be defined. This command must be executed before any new power class values can be defined using `CALCulate:LIMit:ESPectrum:PCLass<class>:MAXimum` and `CALCulate:LIMit:ESPectrum:PCLass<class>:MINimum`.

Parameters:

<NoPowerClasses>	1 to 4 *RST: 1
Example:	`CALC:LIM:ESP:PCL:COUN 2` Two power classes can be defined.

CALCulate:LIMit:ESPectrum:PCLass<class>[:EXCLusive] <State>

This command selects the power class used by the measurement if `CALCulate:LIMit:ESPectrum:MODE` is set to manual.

Note that:

- You can only use power classes for which limits are defined.

Suffix:

<class>	1...4 power class

Parameters:

<State>	ON | OFF *RST: OFF
Example:	`CALC:LIM:ESP:PCL1 ON` Activates the first defined power class.
Manual control:	See "Used Power Classes" on page 89 See "Adding or Removing a Power Class" on page 89

CALCulate:LIMit:ESPectrum:PCLass<class>:LIMit[:STATe] <State>

This command selects the limit check mode for each power class.

Suffix:

<class>	1...4 power class

Parameters:

<State>	**ABSolute** Evaluates only limit lines with absolute power values **RELative** Evaluates only limit lines with relative power values **AND** Evaluates limit lines with relative and absolute power values. A negative result is returned if both limits fail. **OR** Evaluates limit lines with relative and absolute power values. A negative result is returned if at least one limit failed. *RST: REL
Example:	`CALC:LIM:ESP:PCL:LIM ABS`

Manual control: See "Used Power Classes" on page 89

CALCulate:LIMit:ESPectrum:PCLass<class>:MAXimum <Level>

This command defines the upper limit of a particular power class.

Note:

- The last power class always has an upper limit of 200 dBm.
- The upper limit of a power class must always be the same as the lower limit of the subsequent power class.
- The power class must already exist (see CALCulate:LIMit:ESPectrum:PCLass:COUNt on page 502).

Suffix:
<class> 1...4
power class

Parameters:
<Level> Range: -199.9 dBm to 200 dBm

Example: `CALC:LIM:ESP:PCL1:MAX -40 dBm`
Sets the maximum power value of the first power class to -40 dBm.

Manual control: See "PMin/PMax" on page 89

CALCulate:LIMit:ESPectrum:PCLass<class>:MINimum <Level>

This command defines the lower limit of a particular power class.

Note:

- The first power class always has a lower limit of -200 dBm.
- The lower limit of a power class must always be the same as the upper limit of the previous power class.
- The power class must already exist (see CALCulate:LIMit:ESPectrum:PCLass:COUNt on page 502).

Suffix:
<class> 1...4
power class

Parameters:
<Level> Range: -200 dBm to 199.9 dBm

Example: `CALC:LIM:ESP:PCL2:MIN -40 dBm`
Sets the minimum power value of the second power class to -40 dBm.

Manual control: See "PMin/PMax" on page 89

10.3.6.6 Configuring MSR SEM Measurements

The following commands configure MSR SEM measurements. For details see chapter 4.5.4.4, "Multi-Standard Radio (MSR) SEM Measurements", on page 81.

For manual operation see chapter 4.5.5.4, "MSR Settings", on page 89.

[SENSe:]ESPectrum:MSR:APPLy

This command configures the SEM sweep list according to the MSR settings defined by previous commands.

Usage: Event

Manual control: See "Apply to SEM" on page 91

[SENSe:]ESPectrum:MSR:BCATegory <Category>

This command defines the band category for MSR measurements, i.e. the combination of available carriers to measure.

Parameters:

<Category> 1 | 2 | 3

1
2 carriers: LTE FDD and WCDMA

2
3 carriers: LTE FDD, WCDMA and GSM/EDGE

3
2 carriers: LTE TDD and TD-SCDMA

*RST: 1

Manual control: See "Band Category" on page 90

[SENSe:]ESPectrum:MSR:GSM:CPResent <State>

This command defines whether a GSM/Edge carrier is located at the edge of the specified RF bandwidth. In this case, the specification demands specific limits for the SEM ranges.

This command is only available for band category 2 (see `[SENSe:]ESPectrum:MSR:BCATegory` on page 505).

Parameters:

<State> ON | OFF

*RST: OFF

Manual control: See "Carrier Adjacent to RF Bandwidth Edge" on page 90

[SENSe:]ESPectrum:MSR:LTE:CPResent <State>

This command defines whether an LTE FDD 1.4 MHz or 3 MHz carrier is located at the edge of the specified RF bandwidth. In this case, the specification demands specific limits for the SEM ranges.

This command is only available for band category 2 (see `[SENSe:]ESPectrum:MSR:BCATegory` on page 505).

Parameters:

<State>	ON | OFF *RST: OFF

Manual control: See "Carrier Adjacent to RF Bandwidth Edge" on page 90

[SENSe:]ESPectrum:MSR:RFBWidth <Bandwidth>

This command defines the RF bandwidth of the base station for MSR measurements.

Parameters:

<Bandwidth>	Bandwidth in Hz *RST: 10.0 MHz

Manual control: See "Base Station RF Bandwidth" on page 90

10.3.6.7 Configuring the List Evaluation

The following commands configure the list evaluation.

Useful commands for SEM measurements described elsewhere

- `MMEMory:STORe:LIST` on page 696

Remote commands exclusive to SEM measurements

CALCulate<n>:ESPectrum:PSEarch|PEAKsearch:AUTO <State>

This command turns the list evaluation on and off.

Parameters:

<State>	ON | OFF *RST: ON

Example: `CALC:ESP:PSE:AUTO OFF`
Deactivates the list evaluation.

Manual control: See "List Evaluation State" on page 93
See "List Evaluation State" on page 113

CALCulate<n>:ESPectrum:PSEarch|PEAKsearch[:IMMediate]

This command initiates a list evaluation.

Usage: Event

CALCulate<n>:ESPectrum:PSEarch|PEAKsearch:MARGin <Threshold>

This command defines the threshold of the list evaluation.

Parameters:

<Margin>
Range: -200 to 200
*RST: 200
Default unit: dB

Example: `CALC:ESP:PSE:MARG 100`
Sets the margin to 100 dB.

Manual control: See "Margin" on page 93
See "Margin" on page 114

CALCulate<n>:ESPectrum:PSEarch|PEAKsearch:PSHow

This command turns the peak labels in the diagram on and off.

Peak labels are blue squares.

Parameters:

<State> ON | OFF
*RST: OFF

Example: `CALC:ESP:PSE:PSH ON`
Marks all peaks with blue squares.

Manual control: See "Show Peaks" on page 93
See "Show Peaks" on page 113

10.3.6.8 Retrieving Results

The following commands analyze and retrieve measurement results for SEM measurements.

- `CALCulate<n>:LIMit<k>:FAIL` on page 676
- `CALCulate<n>:MARKer<m>:FUNCtion:POWer:RESult?` on page 464
- `TRACe<n>[:DATA]` on page 625
- `TRACe<n>[:DATA]:MEMory?` on page 626
- `TRACe<n>[:DATA]:X?` on page 627

10.3.6.9 Example: SEM Measurement

In this example we will configure and perform an SEM measurement. Note that this example is primarily meant to demonstrate the remote control commands, it does not necessarily reflect a useful measurement task. For most common measurement standards, the R&S FSW performs the measurement optimally with the predefined settings, without further configuration.

```
//------------ Preparing the measurement------------
*RST
//Resets the instrument
SWE:MODE ESP
//Activates SEM Measurement
//SEM has to be in single sweep mode to be configured

INIT:CONT OFF
//Turns on the SEM measurement.

//------------ Managing Measurement Configurations----------
ESP:PRES 'WCDMA\3GPP\UL\3GPP_UL.xml'
//Loads the 3GPP configuration stored in the file '3GPP_UL.xml'

//------------ Defining the Reference Range-----------------
ESP:RRAN?
//Queries the current reference range.
ESP:RTYP CPOW
//Selects the channel power as the power reference.
ESP:BWID 4 MHZ
//Defines a channel bandwidth of 4 MHz for the power reference.
ESP:FILT:RRC ON
ESP:FILT:ALPH 0.5
//Uses an RRC filter with a roll-off factor of 0.5 when measuring
//the reference power.

//------------ Configuring Power Classes--------------------
CALC:LIM:ESP:PCL:COUN 3
//Defines 3 power classes.
CALC:LIM:ESP:LIM -100,0
//Defines the value ranges of the three power classes as [dBm]:
//power class 1: -200 to -100
//power class 2: -100 to 0
//power class 3: 0 to 200
CALC:LIM:ESP:PCL1:LIM ABS
//Defines an absolute limit check for class 1.
CALC:LIM:ESP:PCL2:LIM REL
//Defines a relative limit check for class 2.
CALC:LIM:ESP:MODE MAN
//Defines a manual selection of the power class.
CALC:LIM:ESP:PCL2 ON
//Activates the use of the second power class for the entire measurement.
```

```
//------------ Configuring a Sweep List---------------------
ESP:RANG2:INS AFT
//Inserts a range after range 2.
ESP:RANG1:INS BEF
//Inserts a range before range 1.
ESP:RANG:COUNt?
//Returns the number of measurement ranges in the sweep list (currently 11).
ESP:RANG5:DEL

//Deletes the 11th range.
ESP:RANG1:STOP -10000000

//Defines a stop frequency of -9 MHz for range 1.
ESP:RANG2:STAR -9000000
//Defines a start frequency of -10 MHz for range 2.

ESP:HSP OFF
//Switches off Fast SEM mode so the ranges can be configured individually.

ESP:RANG2:BAND:RES 1000000
//Defines a resolution bandwidth of 1 MHz for range 2.

ESP:RANG2:FILT:TYPE RRC
//Selects an RRC filter for range 2.
ESP:RANG2:BAND:VID 5000000
//Defines a video bandwidth of 5 MHz for range 2.
ESP:RANG2:SWE:TIME 1
//Defines a sweep time of 1 second for range 2.
ESP:RANG2:RLEV 0
//Defines a reference level of 0 dBm for range 2.
ESP:RANG2:INP:ATT 10
//Defines an input attenuation of 10 dB for range 2.


// Create a transducer that can be used.
// It has to cover the corresponding frequency range

SENSe1:CORRection:TRANsducer:SELect 'Transducer'
SENSe1:CORRection:TRANsducer:UNIT 'DB'
SENSe1:CORRection:TRANsducer:COMMent 'Test Transducer'
// Frequency Span 0 MHz bis 20 Ghz
SENSe1:CORRection:TRANsducer:DATA 0e6,5, 20e9,3


ESP:RANG2:TRAN 'Transducer'
//Includes a transducer called 'transducer' for range 2.

//------------ Configuring the limit check------------------
```

```
ESP:RANG:LIM1:STAT AND
//Checks the absolute and relative limits for all ranges in power class 1 and
//fails if both limits are violated. Since power class 2 is set to be used for
//the entire measurement, values for Limit Check 1 are irrelevant. They are
//defined here to demonstrate the use of the MAX function for relative limits.
ESP:RANG2:LIM1:REL:STAR:FUNC MAX
//Enables the use of maximum function for relative limit start. If the value
//exceeds the larger of the absolute (-13 dBm) and relative (-10 dBc) start
//values, the check fails.
ESP:RANG2:LIM1:REL:STAR -10
ESP:RANG2:LIM1:REL:STAR:ABS -13
ESP:RANG2:LIM1:REL:STOP:FUNC MAX
ESP:RANG2:LIM1:REL:STOP -10
ESP:RANG2:LIM1:REL:STOP:ABS -13

ESP:RANG:LIM2:STAT OR
//Checks the absolute and relative limits for all ranges in power class 2 and
//fails if either limit is violated. Since power class 2 is set to be used for
//the entire measurement, values for Limit Check 1 are irrelevant.
ESP:RANG2:LIM2:ABS:STAR 10
ESP:RANG2:LIM2:ABS:STOP 10
//Defines an absolute limit of 10 dBm for the entire range 2 for power class 2.
ESP:RANG2:LIM2:REL:STAR -20
ESP:RANG2:LIM2:REL:STOP -20
//Defines a relative limit of -20 dBc for the entire range 2 for power class 2.

//------------ Configuring List Evaluation-----------------
CALC:ESP:PSE:AUTO ON
//Activates list evaluation, i.e. the peak is determined for each range
//after each sweep.
CALC:ESP:PSE:MARG 10dB
//Defines a peak threshold of 10 dB.

//------------ Managing Measurement Configurations----------

ESP:PRES:STOR 'WCDMA\3GPP\UL\3GPP_UL_User.xml'
//Saves the current configuration in a new file named '3GPP_UL_User'
//in the same directory so the standard is not overwritten.

//------------ Performing the measurement-------------------
INIT:IMM
//One sweep

//------------ Checking the Results-------------------------
CALC:LIM:FAIL?
//Queries the result of the limit check for all ranges.
TRAC:DATA? LIST
//Returns the peak for each range of the SEM measurement as a list.
```

10.3.7 Measuring Spurious Emissions

All remote control commands specific to spurious emissions measurements are described here.

10.3.7.1 Initializing the Measurement

Note that with the R&S FSW, the spurious measurement must be initialized before you can start configuring the sweep list or list evaluation.

INITiate:SPURious

This command initiates a Spurious Emission measurement.

Usage: Event

10.3.7.2 Configuring a Sweep List

The following commands configure the sweep list for spurious emission measurements.

Useful commands for configuring the sweep described elsewhere:

- `[SENSe:]SWEep:MODE` on page 487

Remote commands exclusive to spurious measurements:

[SENSe:]LIST:RANGe<range>:BANDwidth[:RESolution] <RBW>

This command defines the resolution bandwidth for a spurious emission measurement range.

Suffix:
<range> 1...20
Selects the measurement range.

Parameters:
<RBW> Resolution bandwidth.
Refer to the data sheet for available resolution bandwidths.
Default unit: Hz

Manual control: See "RBW" on page 110

[SENSe:]LIST:RANGe<range>:BANDwidth:VIDeo <VBW>

This command defines the video bandwidth for a spurious emission measurement range.

Suffix:
<range> 1...20
Selects the measurement range.

Parameters:
<VBW> Video bandwidth.
Refer to the data sheet for available video bandwidths.
Default unit: Hz

Manual control: See "VBW" on page 110

[SENSe:]LIST:RANGe:BREak <State>

This command controls the sweep.

Parameters:
<State> **ON**
The R&S FSW stops after measuring one range.
To continue with the next range, you have to use `INITiate:CONMeas`.
OFF
The R&S FSW sweeps all ranges in one go.
*RST: OFF

Manual control: See "Stop After Sweep" on page 111

[SENSe:]LIST:RANGe:COUNt?

This command queries the number of ranges in the sweep list.

Return values:

<Ranges> Number of ranges in the sweep list.

Usage: Query only

[SENSe:]LIST:RANGe<range>:DELete

This command removes a range from the sweep list.

Note that

- you cannot delete the reference range
- a minimum of three ranges is mandatory.

Suffix:

<range> 1...20
Selects the measurement range.

Usage: Event

[SENSe:]LIST:RANGe<range>:DETector <Detector>

This command selects the detector for a spurious emission measurement range.

Suffix:

<range> 1...20
Selects the measurement range.

Parameters:

<Detector> **NEGative**
minimum peak detector

POSitive
peak detector

SAMPle
sample detector

RMS
RMS detector

AVERage
average detector

*RST: RMS

Manual control: See "Detector" on page 111

[SENSe:]LIST:RANGe<range>:FILTer:TYPE <FilterType>

This command selects the filter type for a spurious emission measurement range.

Suffix:
<range> 1...20
Selects the measurement range.

Parameters:
<FilterType> **NORMal**
Gaussian filters

CFILter
channel filters

RRC
RRC filters

P5
5 Pole filters

*RST: NORM
The available bandwidths of the filters are specified in the data sheet.

Manual control: See "Filter Type" on page 110

[SENSe:]LIST:RANGe<range>[:FREQuency]:STARt <Frequency>

This command defines the start frequency of a spurious emission measurement range.

Make sure to set an appropriate span. If you set a span that is

- smaller than the span the sweep list covers, the R&S FSW will not measure the ranges that are outside the span - results may be invalid.
- greater than the span the sweep list covers, the R&S FSW will adjust the start frequency of the first range and the stop frequency of the last range to the span

For more information see chapter 4.6, "Spurious Emissions Measurement", on page 104.

Suffix:
<range> 1...20
Selects the measurement range.

Parameters:
<Frequency> Numeric value.
*RST: -12.75 MHz (range 1), -2.515 MHz (range 2), 2.515 MHz (range 3)
Default unit: Hz

Manual control: See "Range Start / Range Stop" on page 110

[SENSe:]LIST:RANGe<range>[:FREQuency]:STOP <Frequency>

This command defines the stop frequency of a spurious emission measurement range.

Make sure to set an appropriate span. If you set a span that is

- smaller than the span the sweep list covers, the R&S FSW will not measure the ranges that are outside the span - results may be invalid.
- greater than the span the sweep list covers, the R&S FSW will adjust the start frequency of the first range and the stop frequency of the last range to the span

For more information seechapter 4.6, "Spurious Emissions Measurement", on page 104 .

Suffix:

<range>	1...20 Selects the measurement range.

Parameters:

<Frequency>	Numeric value. *RST: -2.52 MHz (range 1), 2.52 MHz (range 2), 250.0 MHz (range 3) Default unit: Hz

Manual control: See "Range Start / Range Stop" on page 110

[SENSe:]LIST:RANGe<range>:INPut:ATTenuation <Attenuation>

This command defines the input attenuation for a spurious emission measurement range.

Suffix:

<range>	1...20 Selects the measurement range.

Parameters:

<Attenuation>	Numeric value. Refer to the data sheet for the attenuation range. *RST: 10 dB Default unit: dB

Manual control: See "RF Attenuator" on page 111

[SENSe:]LIST:RANGe<range>:INPut:ATTenuation:AUTO <State>

This command turns automatic selection of the input attenuation for a spurious emission measurement range on and off.

Suffix:

<range>	1...20 Selects the measurement range.

Parameters:

<State>	ON | OFF *RST: ON

Manual control: See "RF Att. Mode" on page 111

[SENSe:]LIST:RANGe<range>:INPut:GAIN:STATe <State>

This command turns the preamplifier for a spurious emission measurement range on and off.

The preamplification is defined by `[SENSe:]LIST:RANGe<range>:INPut:GAIN[:VALue]` on page 516.

Suffix:

<range> 1...20
Selects the measurement range.

Parameters:

<State> ON | OFF
*RST: OFF

Manual control: See "Preamp" on page 111

[SENSe:]LIST:RANGe<range>:INPut:GAIN[:VALue] <Gain>

This command selects the preamplification level for the range.

The command requires option R&S FSW-B24.

Parameters:

<Gain> 15 dB | 30 dB
The availability of preamplification levels depends on the R&S FSW model.
• R&S FSW8: 15dB and 30 dB
• R&S FSW13: 15dB and 30 dB
• R&S FSW26: 30 dB
All other values are rounded to the nearest of these two.
*RST: OFF

[SENSe:]LIST:RANGe<range>:LIMit:STARt <Level>

This command defines an absolute limit for a spurious emission measurement range.

Suffix:

<range> 1...20
Selects the measurement range.

Parameters:

<Level> Absolute limit at the start frequency of a SEM range.
Range: -400 to 400
*RST: 13
Default unit: dBm

Manual control: See "Abs Limit Start/Stop" on page 112

[SENSe:]LIST:RANGe:LIMit:STATe

This command turns the limit check for all spurious emission measurement ranges on and off.

Parameters:

<State> ON | OFF

*RST: OFF

Manual control: See "Limit Check" on page 112

[SENSe:]LIST:RANGe<range>:LIMit:STOP <Level>

This command defines an absolute limit for a spurious emission measurement range.

Suffix:

<range> 1...20

Selects the measurement range.

Parameters:

<Level> Absolute limit at the stop frequency of a SEM range.

Range: -400 to 400

*RST: 13

Default unit: dBm

Manual control: See "Abs Limit Start/Stop" on page 112

[SENSe:]LIST:RANGe<range>:POINts <Points>

This command defines the number of sweep points in a spurious emission measurement range.

Suffix:

<range> 1...20

Selects the measurement range.

Parameters:

<Points> For more information on sweep points see chapter 5.5.1.8, "How Much Data is Measured: Sweep Points and Sweep Count", on page 198.

*RST: 691

Manual control: See "Sweep Points" on page 111

[SENSe:]LIST:RANGe<range>:RLEVel <RefLevel>

This command defines the reference level for a spurious emission measurement range.

Suffix:

<range> 1...20

Selects the measurement range.

Parameters:

<RefLevel>	Reference level. Refer to the data sheet for the reference level range. *RST: 0 dBm
Manual control:	See "Ref. Level" on page 111

[SENSe:]LIST:RANGe<range>:SWEep:TIME <SweepTime>

This command defines the sweep time for a spurious emission measurement range.

Suffix:

<range>	1...20 Selects the measurement range.

Parameters:

<SweepTime>	Sweep time. The range depends on the ratios of the span to the RBW and the RBW to the VBW. Refer to the data sheet for more information.
Manual control:	See "Sweep Time" on page 110

[SENSe:]LIST:RANGe<range>:SWEep:TIME:AUTO <State>

This command turns automatic selection of the sweep time for a spurious emission measurement range on and off.

Suffix:

<range>	1...20 Selects the measurement range.

Parameters:

<State>	ON | OFF *RST: ON
Manual control:	See "Sweep Time Mode" on page 110

[SENSe:]LIST:RANGe<range>:TRANsducer <Transducer>

This command selects a transducer factor for a spurious emission measurement range.

Note that

- the transducer must cover at least the span of the range
- the x-axis has to be linear
- the unit has to be dB

Suffix:

<range>	1...20 Selects the measurement range.

Parameters:

<Transducer>	String containing the transducer file name, including the path information.
Manual control:	See "Transducer" on page 112

10.3.7.3 Configuring the List Evaluation

The following commands configure the list evaluation.

Useful commands for spurious emission measurements described elsewhere

- `MMEMory:STORe:LIST` on page 696

Remote commands exclusive to spurious emission measurements

CALCulate<n>:PSEarch|PEAKsearch:AUTO <State>

This command turns the list evaluation on and off.

Parameters:

<State>	ON | OFF *RST: ON
Example:	`CALC:PSE:AUTO OFF` Deactivates the list evaluation.

CALCulate<n>:PSEarch|PEAKsearch:MARGin <Threshold>

This command defines the threshold of the list evaluation.

Parameters:

<Margin>	Range: -200 to 200 *RST: 200 Default unit: dB
Example:	`CALC:PSE:MARG 100` Sets the threshold to 100 dB.

CALCulate<n>:PSEarch|PEAKsearch:PSHow

This command turns the peak labels in the diagram on and off.

Peak labels are blue squares.

Parameters:

<State> ON | OFF

*RST: OFF

Example: `CALC:PSE:PSH ON`

Marks all peaks with blue squares.

CALCulate<n>:PSEarch|PEAKsearch:SUBRanges <NumberPeaks>

This command defines the number of peaks included in the peak list.

After this number of peaks has been found, the R&S FSW stops the peak search and continues the search in the next measurement range.

Parameters:

<NumberPeaks> Range: 1 to 50

*RST: 25

Example: `CALC:PSE:SUBR 10`

Sets 10 peaks per range to be stored in the list.

Manual control: See "Peaks per Range" on page 114

10.3.7.4 Retrieving and Saving Settings and Results

The following commands analyze and retrieve measurement results for Spurious measurements.

Useful commands for spurious emission measurements described elsewhere

- `CALCulate<n>:LIMit<k>:FAIL` on page 676
- `TRACe<n>[:DATA]` on page 625
- `TRACe<n>[:DATA]:MEMory?` on page 626
- `TRACe<n>[:DATA]:X?` on page 627

10.3.7.5 Programming Example: Spurious Emissions Measurement

In the following example, the Spurious Emissions measurement is configured by defining ranges and parameters to create the following sweep list.

Spurious Emissions

	Range 1	Range 2	Range 3
Range Start	10 MHz	100 MHz	100.0000...
Range Stop	100 MHz	100.0000...	1 GHz
Filter Type	RRC	Normal(3...	Normal(3...
Res BW	24.3 kHz	10 kHz	100 kHz
Video BW	5 MHz	30 kHz	300 kHz
Sweep Time Mode	Manual	Auto	Auto
Sweep Time	1 s	4.01 ms	32.1 ms
Detector	Sample	RMS	RMS
Ref. Level	-20 dBm	0 dBm	0 dBm
RF Att. Mode	Manual	Auto	Auto
RF Attenuator	10 dB	10 dB	10 dB
Preamp	On	Off	Off
Sweep Points	601	4001	32001
Stop After Sweep	Off	Off	Off
Transducer	None	None	None
Limit Check	Absolute	Absolute	Absolute
Abs Limit Start	10 dBm	-13 dBm	-13 dBm
Abs Limit Stop	10 dBm	-13 dBm	-13 dBm

Note that this example is primarily meant to demonstrate the remote control commands, it does not necessarily reflect a useful measurement task.

```
//------------Preparing the measurement-------------
*RST
//Resets the instrument

SWE:MODE LIST
//Activates spurious emissions measurement
//Spurious measurement has to be in single sweep mode to be configured

INIT:CONT OFF
//Selects single sweep mode.

//---------------Configuring a Sweep List----------

LIST:RANG:COUNt?
//Returns the number of measurement ranges in the sweep list.
LIST:RANG4:DEL
//Deletes the fourth range.
LIST:RANG1:STAR 10000000
//Defines a start frequency of 10 MHz for range 1.
LIST:RANG1:STOP 100000000
```

```
//Defines a stop frequency of 100 MHz for range 1.
LIST:RANG1:BAND 500000
//Defines a resolution bandwidth of 500 kHz in range 1.
LIST:RANG1:BAND:VID 5000000
//Defines a video bandwidth of 5 MHz for range 1.
LIST:RANG1:INP:ATT:AUTO OFF
//Turns automatic selection of the input attenuation in range 1 off.

LIST:RANG1:INP:ATT 10
//Defines a input attenuation of 10 dBm for range 1.


LIST:RANG1:FILT:TYPE CFILter
//Selects an Channel filter for range 1.
LIST:RANG1:DET SAMP
//Selects a sample detector for range 1.
LIST:RANG1:POIN 601
//Defines 601 sweep points for range 1.
LIST:RANG1:RLEV -20
//Defines a reference level of -20 dBm for range 1.
LIST:RANG1:SWE:TIME 5
//Defines a manual sweep time of 5 second for range 1.

// Create a transducer that can be used.
// It has to cover the corresponding frequency range
//
SENSe1:CORRection:TRANsducer:SELect 'Test'
SENSe1:CORRection:TRANsducer:UNIT 'DB'
SENSe1:CORRection:TRANsducer:COMMent 'Test Transducer'
// Frequency Span 0 MHz to 20 Ghz
SENSe1:CORRection:TRANsducer:DATA 0e6,5, 20e9,3

SENS:LIST:RANG1:TRAN 'Test'
//Includes a transducer called 'Test' for range 1.

LIST:RANG1:LIM:STAR 10
LIST:RANG1:LIM:STOP 10
//Defines an absolute limit of 10 dBm at the start and stop frequencies of range 1.
LIST:RANG:LIM:STAT ON
//Turns the limit check for all ranges on.

//---------------Configuring the List Evaluation----------
CALC:PSE:MARG 100
//Sets the threshold to 100 dB.
CALC:PSE:PSH ON
//Marks all peaks in the diagram with blue squares.
CALC:PSE:SUBR 10
//Sets 10 peaks per range to be stored in the list.

//--------------Performing the Measurement-----
```

```
INIT:SPUR; *WAI
//Performs a spurious emission measurement and waits until the sweep has finished.

//--------------Retrieving Results-------------
CALC:LIM1:FAIL?
//Queries the result of the check for limit line 1.
TRAC? SPUR
//Queries the peak list of the spurious emission measurement.
```

10.3.8 Analyzing Statistics (APD, CCDF)

All remote control commands specific to statistical measurements are described here.

10.3.8.1 Activating Statistical Measurements

The following commands activate statistical measurements.

CALCulate<n>:STATistics:APD[:STATe] <State>

This command turns the APD measurement on and off.

Parameters:

<State> ON | OFF

*RST: OFF

Example: `CALC:STAT:APD ON`

Switches on the APD measurement.

Manual control: See "APD" on page 33

CALCulate<n>:STATistics:CCDF[:STATe] <State>

This command turns the CCDF on and off.

Parameters:

<State> ON | OFF

*RST: OFF

Example: `CALC:STAT:CCDF ON`

Switches on the CCDF measurement.

Manual control: See "CCDF" on page 33

10.3.8.2 Configuring Statistical Measurements

The following commands configure the measurement.

Useful commands for configuring statistical measurements described elsewhere:

- `[SENSe:]BANDwidth|BWIDth[:RESolution]` on page 568

Remote commands exclusive to statistical measurements:

CALCulate<n>:MARKer<m>:Y:PERCent <Probability>

This command sets a marker to a particular probability value. You can query the corresponding level with `CALCulate<n>:MARKer<m>:X`.

Using the command turns delta markers into normal markers.

This command is available for CCDF measurements.

Parameters:

<Probability> Range: 0 % to 100 %

Example: `CALC1:MARK:Y:PERC 95PCT`
Positions marker 1 to a probability of 95 %.

Manual control: See "Percent Marker (CCDF only)" on page 123

CALCulate<n>:STATistics:NSAMples <Samples>

This command defines the number of samples included in the analysis of statistical measurement functions.

Parameters:

<Samples> Range: Min: 100, Max: depends on the RBW filter
*RST: 100000

Example: `CALC:STAT:NSAM 500`
Sets the number of measurement points to be acquired to 500.

Manual control: See "Number of Samples" on page 123

10.3.8.3 Using Gate Ranges for Statistical Measurements

The following commands control gated statistical measurements.

[SENSe:]SWEep:EGATe:TRACe<k>:COMMent <Comment>

This command defines a comment for the gate of a particular trace.

Parameters:

<Comment>	String containing the comment.
Example:	`SWE:EGAT:TRAC1:COMM 'MyComment'` Defines a comment for the gate in trace 1.
Manual control:	See "Comment" on page 125

[SENSe:]SWEep:EGATe:TRACe:PERiod <Length>

This command defines the length of the gate for all traces.

The gate length applies to all traces.

Parameters:

<Length>	Range: 100 ns to 1000 s *RST: 2 ms
Example:	`SWE:EGAT:TRAC1:PER 5ms` Defines the period for gated triggering to 5 ms.
Manual control:	See "Period" on page 125

[SENSe:]SWEep:EGATe:TRACe<k>:STARt<range> <Time>

This command defines the start time for a gate range.

Suffix:

<range>	1...3 gate range

Parameters:

<Time>	The value range depends on the gate period you have set for the selected trace with `[SENSe:]SWEep:EGATe:TRACe:PERiod`. The following rules apply: • the start time may not be higher than the length of the gate • the start time may not be lower than the stop time of the gate range of a lower order The reset values depend on the gate range. • for gate range 1, the start time is 0 ms • for gate range 3, the start time is 2 ms • for gate range 5, the start time is 4 ms

Example: `SWE:EGAT:TRAC1:STAR1 3ms`
Sets the Starting point for range 1 on trace 1 at 3 ms.

Manual control: See "Range <x> Start/Stop" on page 125

[SENSe:]SWEep:EGATe:TRACe<k>[:STATe<range>] <State>

This command includes or excludes a gate range for a particular trace.

Suffix:
<range> 1...3
gate range

Parameters:
<State> ON | OFF
*RST: OFF

Example: `SWE:EGAT:TRAC1:STAT1 ON`
Activates gate range 1 for trace 1.

Manual control: See "Gated Trigger" on page 124
See "Range <x> Use" on page 125

[SENSe:]SWEep:EGATe:TRACe<k>:STOP<range> <Time>

This command defines the stop time for a gate range.

Suffix:
<range> 1...3
gate range

Parameters:
<Time> The value range depends on the gate period you have set for the selected trace with `[SENSe:]SWEep:EGATe:TRACe:PERiod`.
The following rules apply:
• the stop time may not be higher than the length of the gate
• the stop time may not be lower than the start time
The reset values depend on the gate range.
• for gate range 1, the stop time is 1 ms
• for gate range 3, the stop time is 3 ms
• for gate range 5, the stop time is 5 ms

Example: `SWE:EGAT:TRAC1:STOP1 5ms`
Sets the stopping point for range 1 on trace 1 at 5 ms.

Manual control: See "Range <x> Start/Stop" on page 125

10.3.8.4 Scaling the Diagram

The following commands set up the diagram for statistical measurements.

CALCulate<n>:STATistics:PRESet

This command resets the scale of the diagram (x- and y-axis).

- Reference level (x-axis)
  0.0 dBm
- Display range (x-axis) for APD measurements
  100 dB
- Display range (x-axis) for CCDF measurements
  20 dB
- Upper limit of the y-axis
  1.0
- Lower limit of the y-axis
  1E-6

Example: `CALC:STAT:PRES`
Resets the scaling for statistical functions

Usage: Event

Manual control: See "Default Settings" on page 127

CALCulate<n>:STATistics:SCALe:AUTO ONCE

This command initiates an automatic scaling of the diagram (x- and y-axis).

To obtain maximum resolution, the level range is set as a function of the measured spacing between peak power and the minimum power for the APD measurement and of the spacing between peak power and mean power for the CCDF measurement. In addition, the probability scale for the number of test points is adapted.

To get valid results, you have to perform a complete sweep with synchronization to the end of the auto range process. This is only possible in single sweep mode.

Parameters:
ONCE

Example: `CALC:STAT:SCAL:AUTO ONCE;*WAI`
Adapts the level setting for statistical measurements.

Usage: Event

Manual control: See "Adjust Settings" on page 124

CALCulate<n>:STATistics:SCALe:X:RANGe <Range>

This command defines the display range of the x-axis for statistical measurements. The effects are identical to `DISPlay[:WINDow<n>]:TRACe:Y[:SCALe]`.

Parameters:

<Range> Range: 1 dB to 200 dB
*RST: 100 dB

Example: `CALC:STAT:SCAL:X:RANG 20dB`

Manual control: See "X-Axis" on page 127
See "Range" on page 127

CALCulate<n>:STATistics:SCALe:X:RLEVel <RefLevel>

This command sets the reference level for statistical measurements. The effects are identical to `DISPlay[:WINDow<n>]:TRACe:Y[:SCALe]:RLEVel`.

Note that in case of statistical measurements the reference level applies to the x-axis.

Parameters:

<RefLevel> The unit is variable.
If a reference level offset is included, the range is adjusted by that offset.
Range: -130 dBm to 30 dBm
*RST: 0 dBm

Example: `CALC:STAT:SCAL:X:RLEV -60dBm`

Manual control: See "X-Axis" on page 127
See "Ref Level" on page 127

CALCulate<n>:STATistics:SCALe:Y:LOWer <Magnitude>

This command defines the lower vertical limit of the diagram.

Parameters:

<Magnitude> The number is a statistical value and therefore dimensionless.
Range: 1E-9 to 0.1
*RST: 1E-6

Example: `CALC:STAT:SCAL:Y:LOW 0.001`

Manual control: See "Y-Axis" on page 127
See "Y-Max / Y-Min" on page 127

CALCulate<n>:STATistics:SCALe:Y:UNIT <Unit>

This command selects the unit of the y-axis.

Parameters:

<Unit> PCT | ABS

*RST: ABS

Example: `CALC:STAT:SCAL:Y:UNIT PCT`
Sets the percentage scale.

Manual control: See "Y-Axis" on page 127
See "Y-Unit" on page 127

CALCulate<n>:STATistics:SCALe:Y:UPPer <Magnitude>

This command defines the upper vertical limit of the diagram.

Parameters:

<Magnitude> The number is a statistical value and therefore dimensionless.

Range: 1E-5 to 1.0
*RST: 1.0

Example: `CALC:STAT:SCAL:Y:UPP 0.01`

Manual control: See "Y-Axis" on page 127
See "Y-Max / Y-Min" on page 127

10.3.8.5 Retrieving Results

The following commands are required to retrieve the measurement results.

Useful commands for retrieving results described elsewhere:

- `CALCulate<n>:MARKer<m>:X` on page 632

Remote commands exclusive to statistical results

CALCulate<n>:STATistics:CCDF:X<t>? <Probability>

This command queries the results of the CCDF.

Query parameters:

<Probability>

P0_01
Level value for 0.01 % probability

P0_1
Level value for 0.1 % probability

P1
P1: Level value for 1 % probability

P10
Level value for 10 % probability

Example: `CALC:STAT:CCDF:X1? P10`
Returns the level values that are over 10 % above the mean value.

Usage: Query only

CALCulate<n>:STATistics:RESult<t> <ResultType>

This command queries the results of a CCDF or ADP measurement.

Parameters:

<ResultType>

MEAN
Average (=RMS) power in dBm measured during the measurement time.

PEAK
Peak power in dBm measured during the measurement time.

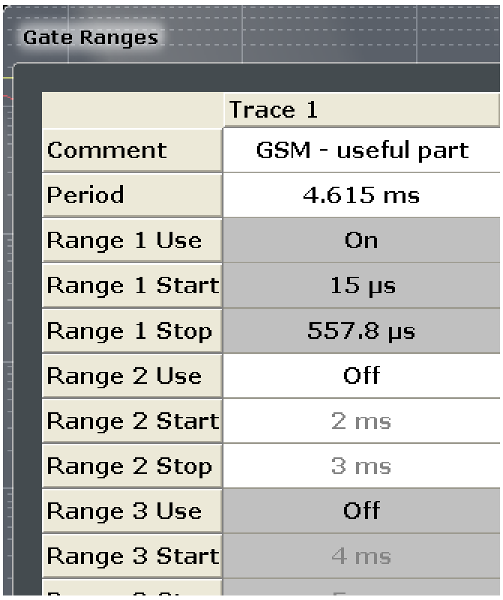
CFACtor
Determined crest factor (= ratio of peak power to average power) in dB.

ALL
Results of all three measurements mentioned before, separated by commas: <mean power>,<peak power>,<crest factor>

Example: `CALC:STAT:RES2? ALL`
Reads out the three measurement results of trace 2. Example of answer string: 5.56,19.25,13.69 i.e. mean power: 5.56 dBm, peak power 19.25 dBm, crest factor 13.69 dB

10.3.8.6 Programming Example: Measuring Statistics

This example demonstrates how to determine statistical values for a measurement in a remote environment using the gated statistics example described in chapter 4.7.4, "APD and CCDF Basics - Gated Triggering", on page 121.



```
//-----------Configuring the measurement ------------
*RST
//Reset the instrument
TRIG:SOUR EXT
//Defines the use of an external trigger.
TRIG:HOLD 25us
//Defines a trigger offset of 25 µs.
CALC:STAT:APD ON
//Activates APD measurement.
CALC:STAT:NSAM 1000
//Sets the number of samples to be included in the statistical evaluation to 1000.

//-------------Defining Gate ranges -----------------

SWE:EGAT:TRAC1:COMM 'GSM - useful part'
//Defines a comment for the gate
SWE:EGAT:TRAC1:PER 4.61536ms
//Sets the gate period to 4.61536ms.
SWE:EGAT:TRAC1:STAR1 15us
//Sets the start of range 1 to 15 µs.
SWE:EGAT:TRAC1:STOP1 557.8us
//Sets the end of range 1 to 15 µs (start time) + 542.77 µs (useful part) = 557.8 µs.
SWE:EGAT:TRAC1:STAT1 ON
//Activates the use of range 1.

//--------------Performing the Measurement-----
INIT:CONT OFF
//Selects single sweep mode.
INIT;*WAI
//Initiates a new measurement and waits until the sweep has finished.

//---------------Retrieving Results-------------
CALC:STAT:RES1? MEAN
//Returns the mean average power for the useful part of the GSM signal.

//------------- Determining the CCDF values-------------------

CALC:STAT:CCDF ON
//Activates CCDF measurement.
CALC:MARK2:Y:PERC 95PCT
//Sets marker 2 to the 95% probability value.
INIT;*WAI
//Initiates a new measurement and waits until the sweep has finished.
CALC:STAT:CCDF:X? P1
//Returns the level value for 10% probability for the CCDF.
CALC:MARK2:X?
//Returns the level for a probability of 95%.
```

10.3.9 Measuring the Time Domain Power

All remote control commands specific to time domain power measurements are described here.

10.3.9.1 Configuring the Measurement

The following remote commands measure the time domain power.

Useful commands for time domain power measurements described elsewhere

- `CALCulate:MARKer:X:SLIMits:LEFT`
- `CALCulate:MARKer:X:SLIMits:RIGHT`
- `CALCulate:MARKer:X:SLIMits[:STATe]`

Remote commands exclusive to time domain power measurements

CALCulate<n>:MARKer<m>:FUNCtion:SUMMary:AOFF

This command turns all time domain power evaluation modes off.

Usage: Event

CALCulate<n>:MARKer<m>:FUNCtion:SUMMary:AVERage <State>

This command switches on or off averaging for the active power measurement in zero span in the window specified by the suffix <n>. If activated, a time domain value is calculated from the trace after each sweep; in the end, all values are averaged to calculate the final result.

The number of results required for the calculation of average is defined with `[SENSe:]AVERage<n>:COUNt` .

Averaging is reset by switching it off and on again.

Synchronization to the end of averaging is only possible in single sweep mode.

Parameters:

<State> ON | OFF

*RST: OFF

Example:

```
INIT:CONT OFF
```
Switches to single sweep mode.
```
CALC:MARK:FUNC:SUMM:AVER ON
```
Switches on the calculation of average.
```
AVER:COUN 200
```
Sets the measurement counter to 200.
```
INIT;*WAI
```
Starts a sweep and waits for the end.

CALCulate<n>:MARKer<m>:FUNCtion:SUMMary:PHOLd <State>

This command switches on or off the peak-hold function for the active power measurement in zero span in the window specified by the suffix <n>. If activated, the peak for each sweep is compared to the previously stored peak; the maximum of the two is stored as the current peak.

The peak-hold function is reset by switching it off and on again.

Parameters:

<State> ON | OFF

*RST: OFF

CALCulate<n>:MARKer<m>:FUNCtion:SUMMary[:STATe] <State>

This command turns time domain power measurements on and off. This measurement in only available in zero span.

When you turn the measurement on, the R&S FSW activates a marker and positions it on the peak power level in the marker search range.

Parameters:

<State> ON | OFF

*RST: OFF

Manual control: See "Time Domain Power" on page 32

10.3.9.2 Retrieving Measurement Results

The following commands query the results for time domain measurements.

Measuring the Mean Power

CALCulate<n>:MARKer<m>:FUNCtion:SUMMary:MEAN:AVERage:RESult?

This command queries the average mean time domain power. The query is only possible if averaging has been activated previously using `CALCulate<n>:MARKer<m>:FUNCtion:SUMMary:AVERage` on page 532.

To get a valid result, you have to perform a complete measurement with synchronization to the end of the measurement before reading out the result. This is only possible for single sweeps. See also `INITiate:CONTinuous` on page 460.

Return values:

<MeanPower>	Mean power of the signal during the measurement time.
Usage:	Query only

CALCulate<n>:MARKer<m>:FUNCtion:SUMMary:MEAN:PHOLd:RESult?

This command queries the maximum mean time domain power. The query is only possible if the peak hold function has been activated previously using `CALCulate<n>:MARKer<m>:FUNCtion:SUMMary:PHOLd`.

To get a valid result, you have to perform a complete measurement with synchronization to the end of the measurement before reading out the result. This is only possible for single sweeps. See also `INITiate:CONTinuous` on page 460.

Return values:

<MeanPower>	Mean power of the signal during the measurement time.
Usage:	Query only

CALCulate<n>:MARKer<m>:FUNCtion:SUMMary:MEAN:RESult?

This command queries the mean time domain power.

To get a valid result, you have to perform a complete measurement with synchronization to the end of the measurement before reading out the result. This is only possible for single sweeps. See also `INITiate:CONTinuous` on page 460.

Return values:

<MeanPower>	Mean power of the signal during the measurement time.
Usage:	Query only
Manual control:	See "Results" on page 133

CALCulate<n>:MARKer<m>:FUNCtion:SUMMary:MEAN[:STATe] <State>

This command turns the evaluation to determine the mean time domain power on and off.

The R&S FSW performs the measurement on the trace marker 1 is positioned on.

Parameters:

<State>	ON | OFF	
	*RST:	OFF

Manual control: See "Results" on page 133

Measuring the Peak Power

CALCulate<n>:MARKer<m>:FUNCtion:SUMMary:PPEak:AVERage:RESult?

This command queries the average positive peak time domain power. The query is only possible if averaging has been activated previously using `CALCulate<n>:MARKer<m>:FUNCtion:SUMMary:AVERage` on page 532.

To get a valid result, you have to perform a complete measurement with synchronization to the end of the measurement before reading out the result. This is only possible for single sweeps. See also `INITiate:CONTinuous` on page 460.

Parameters:

<PeakPower> Peak power of the signal during the measurement time.

Usage: Query only

CALCulate<n>:MARKer<m>:FUNCtion:SUMMary:PPEak:PHOLd:RESult?

This command queries the maximum positive peak time domain power. The query is only possible if the peak hold function has been activated previously using `CALCulate<n>:MARKer<m>:FUNCtion:SUMMary:PHOLd`.

To get a valid result, you have to perform a complete measurement with synchronization to the end of the measurement before reading out the result. This is only possible for single sweeps. See also `INITiate:CONTinuous` on page 460.

Parameters:

<PeakPower> Peak power of the signal during the measurement time.

Usage: Query only

CALCulate<n>:MARKer<m>:FUNCtion:SUMMary:PPEak:RESult?

This command queries the positive peak time domain power.

To get a valid result, you have to perform a complete measurement with synchronization to the end of the measurement before reading out the result. This is only possible for single sweeps. See also `INITiate:CONTinuous` on page 460.

Parameters:

<PeakPower> Peak power of the signal during the measurement time.

Usage: Query only

Manual control: See "Results" on page 133

CALCulate<n>:MARKer<m>:FUNCtion:SUMMary:PPEak[:STATe] <State>

This command turns the evaluation to determine the positive peak time domain power on and off.

The R&S FSW performs the measurement on the trace marker 1 is positioned on.

Parameters:

<State>	ON | OFF *RST: OFF
Manual control:	See "Results" on page 133

Measuring the RMS Power

CALCulate<n>:MARKer<m>:FUNCtion:SUMMary:RMS:AVERage:RESult?

This command queries the average RMS of the time domain power. The query is only possible if averaging has been activated previously using `CALCulate<n>:MARKer<m>:FUNCtion:SUMMary:AVERage` on page 532.

To get a valid result, you have to perform a complete measurement with synchronization to the end of the measurement before reading out the result. This is only possible for single sweeps. See also `INITiate:CONTinuous` on page 460.

Return values:

<RMSPower>	RMS power of the signal during the measurement time.
Usage:	Query only

CALCulate<n>:MARKer<m>:FUNCtion:SUMMary:RMS:PHOLd:RESult?

This command queries the maximum RMS of the time domain power. The query is only possible if the peak hold function has been activated previously using `CALCulate<n>:MARKer<m>:FUNCtion:SUMMary:PHOLd`.

To get a valid result, you have to perform a complete measurement with synchronization to the end of the measurement before reading out the result. This is only possible for single sweeps. See also `INITiate:CONTinuous` on page 460.

Return values:

<RMSPower>	RMS power of the signal during the measurement time.
Usage:	Query only

CALCulate<n>:MARKer<m>:FUNCtion:SUMMary:RMS:RESult?

This command queries the RMS of the time domain power.

To get a valid result, you have to perform a complete measurement with synchronization to the end of the measurement before reading out the result. This is only possible for single sweeps. See also `INITiate:CONTinuous` on page 460.

Return values:

<RMSPower> RMS power of the signal during the measurement time.

Usage: Query only

Manual control: See "Results" on page 133

CALCulate<n>:MARKer<m>:FUNCtion:SUMMary:RMS[:STATe] <State>

This command turns the evaluation to determine the RMS time domain power on and off.

The R&S FSW performs the measurement on the trace marker 1 is positioned on.

Parameters:

<State> ON | OFF

*RST: OFF

Manual control: See "Results" on page 133

Measuring the Standard Deviation

CALCulate<n>:MARKer<m>:FUNCtion:SUMMary:SDEViation:AVERage:RESult?

This command queries the average standard deviation of the time domain power. The query is only possible if averaging has been activated previously using `CALCulate<n>:MARKer<m>:FUNCtion:SUMMary:AVERage` on page 532.

To get a valid result, you have to perform a complete measurement with synchronization to the end of the measurement before reading out the result. This is only possible for single sweeps. See also `INITiate:CONTinuous` on page 460.

Return values:

<StandardDeviation> Standard deviation of the signal during the measurement time.

Usage: Query only

CALCulate<n>:MARKer<m>:FUNCtion:SUMMary:SDEViation:PHOLd:RESult?

This command queries the maximum standard deviation of the time domain power. The query is only possible if the peak hold function has been activated previously using `CALCulate<n>:MARKer<m>:FUNCtion:SUMMary:PHOLd`.

To get a valid result, you have to perform a complete measurement with synchronization to the end of the measurement before reading out the result. This is only possible for single sweeps. See also `INITiate:CONTinuous` on page 460.

Return values:

<StandardDeviation> Standard deviation of the signal during the measurement time.

Usage: Query only

CALCulate<n>:MARKer<m>:FUNCtion:SUMMary:SDEViation:RESult?

This command queries the standard deviation of the time domain power.

To get a valid result, you have to perform a complete measurement with synchronization to the end of the measurement before reading out the result. This is only possible for single sweeps. See also `INITiate:CONTinuous` on page 460.

Return values:

<StandardDeviation> Standard deviation of the signal during the measurement time.

Usage: Query only

CALCulate<n>:MARKer<m>:FUNCtion:SUMMary:SDEViation[:STATe] <State>

This command turns the evaluation to determine the standard deviation of the time domain power on and off.

The R&S FSW performs the measurement on the trace marker 1 is positioned on.

Parameters:

<State> ON | OFF

*RST: OFF

10.3.9.3 Programming Example: Time Domain Power

This programming example demonstrates the measurement example described in chapter 4.8.6, "Measurement Example", on page 135 in a remote environment.

```
//-------------Configuring the Measurement-----------------------
*RST
//Resets the instrument

INIT:CONT OFF
//Turns on single sweep mode.

FREQ:CENT 1.8GHz
//Sets the center frequency to 1.8 GHz.

BAND:RES 100kHz
//Sets the bandwidth to 100 kHz.

SWE:TIME 10ms
//Sets the sweep time to 640 µs.

FREQ:SPAN 0
//Sets the instrument to zero span.


CALC:MARK:FUNC:SUMM:STAT ON
//Turns on time domain power measurements.
```

```
CALC:MARK:FUNC:SUMM:MEAN ON
CALC:MARK:FUNC:SUMM:PPE ON
CALC:MARK:FUNC:SUMM:RMS ON
//Turns the evalution of the mean, peak and RMS time domain power.

CALC:MARK:X:SLIM ON
//Activates limit lines for evaluation.

CALC:MARK:X:SLIM:LEFT 1ms
//Sets the left limit line to 326 µs.

CALC:MARK:X:SLIM:RIGH 6ms
//Sets the right limit line to 538 µs.

//--------------Performing the Measurement------------------------

INIT;*WAI
//Initiates the measurement and waits until the measurement is finished.

//-------------Retrieving the Results--------------------------------
CALC:MARK:FUNC:SUMM:MEAN:RES?
CALC:MARK:FUNC:SUMM:PPE:RES?
CALC:MARK:FUNC:SUMM:RMS:RES?
//Queries the mean, peak and RMS time domain power.
```

10.3.10 Measuring the Harmonic Distortion

All remote control commands specific to harmonic distortion measurements are described here.

10.3.10.1 Activating the Measurement

The following command activates harmonic distortion measurement.

CALCulate<n>:MARKer<m>:FUNCtion:HARMonics[:STATe] <State>

This command turns the harmonic distortion measurement on and off.

Note the following:

- If you perform the measurement in the frequency domain, the search range for the frequency of the first harmonic, whose power is determined, is defined by the last span.
- If you perform the measurement in the time domain, the current center frequency is used as the frequency of the first harmonic. Thus, the frequency search is bypassed. The first harmonic frequency is set by a specific center frequency in zero span before the harmonic measurement is started.

Parameters:

<State> ON | OFF

*RST: OFF

Example: `CALC:MARK:FUNC:HARM ON`
Activates the harmonic distortion measurement.

10.3.10.2 Configuring the Measurement

The following commands control the harmonic distortion measurement.

Useful commands for harmonic distortion measurements described elsewhere

- `CALCulate<n>:MARKer<m>:FUNCtion:CENTer` on page 563
- `[SENSe:]SWEep:TIME:AUTO` on page 573

Remote commands exclusive to harmonic distortion measurements

---

CALCulate<n>:MARKer<m>:FUNCtion:HARMonics:BANDwidth:AUTO <State>

This command selects the resolution bandwidth of the harmonic in respect to the bandwidth of the first harmonic.

Parameters:

<State> OFF | ON

OFF
identical

ON
a multiple

*RST: ON

Manual control: See "Harmonic RBW Auto" on page 141

CALCulate<n>:MARKer<m>:FUNCtion:HARMonics:NHARmonics <NoHarmonics>

This command selects the number of harmonics that the R&S FSW looks for.

Parameters:

<NoHarmonics>	Range:	1 to 26
	*RST:	10

Manual control: See "No. of Harmonics" on page 140

CALCulate<n>:MARKer<m>:FUNCtion:HARMonics:PRESet

This command initiates a measurement to determine the ideal configuration for the harmonic distortion measurement.

The method depends on the span.

- Frequency domain (span > 0)
  Frequency and level of the first harmonic are determined and used for the measurement list.
- Time domain (span = 0)
  The level of the first harmonic is determined. The frequency remains unchanged.

Usage: Event

Manual control: See "Adjust Settings" on page 141

10.3.10.3 Retrieving Results

The following commands retrieve the results of the harmonic distortion measurement.

CALCulate<n>:MARKer<m>:FUNCtion:HARMonics:DISTortion? TOTal

This command queries the total harmonic distortion of the signal.

To get a valid result, you have to perform a complete measurement with synchronization to the end of the measurement before reading out the result. This is only possible for single sweeps. See also `INITiate:CONTinuous` on page 460.

Return values:

<Result> <Distortion_%>,<Distortion_dB>

Pair of values, one showing the THD in %, one in dB.

Usage: Query only

CALCulate<n>:MARKer<m>:FUNCtion:HARMonics:LIST?

This command queries the position of the harmonics.

To get a valid result, you have to perform a complete measurement with synchronization to the end of the measurement before reading out the result. This is only possible for single sweeps. See also `INITiate:CONTinuous` on page 460.

Return values:

<Harmonics>	Returns one value for every harmonic. The first value is the absolute power of the first harmonic. The unit is variable. The other values are power levels relative to the first harmonic. The unit for these is dB.
Usage:	Query only

10.3.10.4 Example: Measuring the Harmonic Distortion

```
//-------------Configuring the Measurement-------------
*RST
//Resets the instrument.

INIT:CONT OFF
//Turns on single sweep mode.

CALC:MARK:FUNC:HARM ON
//Turns on the harmonic distortion measurement.
CALC:MARK:FUNC:HARM:NHAR 3
//Defines three harmonics to be found.

CALC:MARK:FUNC:HARM:BAND:AUTO OFF
//Turns off automatic bandwidth selection.
CALC:MARK:FUNC:HARM:PRES
//Determines the ideal configuration.

//-------------Performing the Measurement------------------

INIT;*WAI
//Initiates the measurement and finishes the sweep.

//-------------Retrieving the Results------------------------
CALC:MARK:FUNC:HARM:LIST?
//Queries the position of the harmonics.
CALC:MARK:FUNC:HARM:DIST? TOT
//Queries the total harmonic distortion.
```

10.3.11 Measuring the Third Order Intercept Point

10.3.11.1 Determining the TOI

All remote control commands specific to TOI measurements are described here.

Useful commands for TOI measurements described elsewhere

- `CALCulate<n>:DELTamarker<m>:X` on page 630
- `CALCulate<n>:DELTamarker<m>:X:RELative?` on page 642
- `CALCulate<n>:DELTamarker<m>:Y?` on page 642
- `CALCulate<n>:MARKer<m>:X` on page 632
- `CALCulate<n>:MARKer<m>:Y?` on page 643

Remote commands exclusive to TOI measurements

CALCulate<n>:MARKer<m>:FUNCtion:TOI[:STATe] <State>

This command initiates a measurement to determine the third intercept point.

A two-tone signal with equal carrier levels is expected at the RF input of the instrument. Marker 1 and marker 2 (both normal markers) are set to the maximum of the two signals. Delta marker 3 and delta marker 4 are positioned to the intermodulation products. The delta markers can be modified separately afterwards with the `CALCulate<n>:DELTamarker<m>:X` command.

The third-order intercept is calculated from the level spacing between the normal markers and the delta markers.

Parameters:

<State> ON | OFF
*RST: OFF

Example: `CALC:MARK:FUNC:TOI ON`
Switches on the measurement of the third-order intercept.

Manual control: See "TOI" on page 33

CALCulate:MARKer:FUNCtion:TOI:SEARchsignal ONCE

This command initiates a search for signals in the current trace to determine the third intercept point.

Parameters:

ONCE

Example: `CALC:MARK:FUNC:TOI:SEAR ONCE`
Executes the search for 2 signals and their intermodulation product at the currently available trace.

Usage: Event

Manual control: See "Search Signals" on page 148

CALCulate<n>:MARKer<m>:FUNCtion:TOI:RESult?

This command queries the results for the third order intercept point measurement.

To get a valid result, you have to perform a complete measurement with synchronization to the end of the measurement before reading out the result. This is only possible for single sweeps. See also `INITiate:CONTinuous` on page 460.

Parameters:

<TOI> Third order intercept point.

Example:

`INIT:CONT OFF`
Switches to single sweep mode.
`CALC:MARK:FUNC:TOI ON`
Switches the intercept measurement.
`INIT;*WAI`
Starts a sweep and waits for the end.
`CALC:MARK:FUNC:TOI:RES?`
Outputs the measured value.

Usage: Query only

Manual control: See "TOI" on page 33

10.3.11.2 Programming Example: Measuring the TOI

This example demonstrates how to determine the TOI in a remote environment.

```
//-----------Configuring the measurement ------------
*RST
//Reset the instrument
CALC:MARK:FUNC:TOI ON
//Activate TOI measurement.

//--------------Performing the Measurement-----
INIT:CONT OFF
//Selects single sweep mode.

CALC:MARK:FUNC:TOI:SEAR ONCE
//Initiates a search for signals in the current trace.

//---------------Retrieving Results-------------
CALC:MARK:FUNC:TOI:RES?
//Returns the TOI.
```

10.3.12 Measuring the AM Modulation Depth

All remote control commands specific to AM modulation depth measurements are described here.

10.3.12.1 Configuring the Measurement

The following commands control the measurement.

Useful commands for AM modulation depth described elsewhere

- CALCulate<n>:DELTamarker<m>:X on page 630
- CALCulate<n>:DELTamarker<m>:X:RELative? on page 642
- CALCulate<n>:MARKer<m>:X on page 632

Remote commands exclusive to AM modulation depth measurements

---

CALCulate<n>:MARKer<m>:FUNCtion:MDEPth[:STATe] <State>

This command turns the AM Modulation Depth measurement on and off.

To work correctly, the measurement requires an AM modulated signal.

Parameters:
<State> ON | OFF
*RST: OFF

Manual control: See "AM Mod Depth" on page 33

CALCulate<n>:MARKer<m>:FUNCtion:MDEPth:SEARchsignal ONCE

This command initiates a search for the signals required for the AM depth measurement.

Note that the command does not perform a new measurement, but looks for the signals on the current trace.

Parameters:
ONCE

Example: `CALC:MARK:FUNC:MDEP:SEAR ONCE`
Executes the search of an AM modulated signal at the currently available trace.

Usage: Event

Manual control: See "Search Signals" on page 153

CALCulate<n>:MARKer<m>:FUNCtion:MDEPth:RESult?

This command queries the results of the AM modulation depth measurement..

To get a valid result, you have to perform a complete measurement with synchronization to the end of the measurement before reading out the result. This is only possible for single sweeps. See also INITiate:CONTinuous on page 460.

Return values:

<ModulationDepth> Modulation depth in %.

Usage: Query only

Manual control: See "AM Mod Depth" on page 33

10.3.12.2 Example: Measuring the AM Modulation Depth

This example demonstrates how to determine the AM modulation depth in a remote environment. Note that without a real input signal this measurement will not return useful results.

```
//-----------Configuring the measurement ------------
*RST
//Reset the instrument
FREQ:CENT 100MHz
//Set center frequency
FREQ:SPAN 10KHz
// Set span
CALC:MARK:FUNC:MDEP ON
//Activate AM modulation depth measurement.

//--------------Performing the Measurement-----
INIT:CONT OFF
//Selects single sweep mode.
INIT:IMM
// Perform a single measurement
CALC:MARK:FUNC:MDEP:SEAR ONCE
//Initiates a search for signals in the current trace.

//---------------Retrieving Results-------------
CALC:MARK:FUNC:MDEP:RES?
//Queries the measurement results.

//If the results are not accurate, change the position of the
//the temporary markers manually.

//----Changing the position of the temp markers-----
CALC:MARK:X 100MHZ
//Positions the reference marker on 100 MHz.
CALC:DELT2:X 5KHZ
//Positions delta marker 2 and 3 at a distance of 5 kHz to the reference marker.
CALC:DELT3:X 1KHZ
//Corrects the position of delta marker 3 by 1 kHz.

CALC:MARK:FUNC:MDEP:RES?
//Queries the measurement results for the repositioned markers.
```

10.3.13 List Evaluations

A list evaluation is a mutliple power measurement that measures the power at up to 200 frequencies. The measurement itself is a time domain measurement. Note that if you set a span greater than 0, the R&S FSW aborts the list evaluation.

Noise cancellation in list evaluations

As of R&S FSW firmware version 1.30, noise cancellation is also available in zero span and thus also for list evaluations. See "Noise cancellation" on page 47 for details.

List evaluations allow for a different instrument setup for each frequency you want to measure. You can define most of the settings with the commands described here. Settings not covered by the commands listed below can be controlled with the common commands (see chapter 10.5, "Setting Basic Measurement Parameters", on page 562. Note that these commands have to be sent prior to the commands that control the list evaluation.

In case of a triggered measurement, a separate trigger event is required for each frequency to initiate that measurement. Note that you have to make changes to the trigger level in the time domain in order for it to take effect for the List Evaluation commands.

The list evaluation is incompatible to other measurement functions (e.g. marker functionality or statistics). If you use a command that controls those functions, the R&S FSW aborts the list evaluation.

The R&S FSW also aborts the list evaluation if you end the remote session.

The commands can be used in two different ways.

- Instrument setup, measurement and querying of the results in a single command line. This method causes the least delay between the measurement and the result output. However, it requires the control computer to wait for the response from the instrument.
- Instrument setup and querying of the result list at the end of the measurement: With this method, the control computer may be used for other activities while the measurement is being performed. However, more time is needed for synchronization via service request.

10.3.13.1 Performing List Evaluations

All remote control commands specific to list evaluations (which are available via remote control only) are described here.

Useful commands for list evaluation described elsewhere:

- `[SENSe:]POWer:NCORrection` on page 576

Remote commands exclusive to list evaluation

[SENSe:]LIST:POWer:RESult?

This command queries the results of the list evaluation.

This command may be used to obtain measurement results in an asynchronous way, using the service request mechanism for synchronization to the end of the measurement.

If there are no results, the command returns an error.

Return values:

<PowerLevel> Power level for each frequency included in the measurement.
The command returns up to 3 power levels for each frequency, depending on the number of evaluation modes you have turned on with `[SENSe:]LIST:POWer:SET`.
The result is a list of floating point values separated by commas.
The unit depends on `CALCulate<n>:UNIT:POWer`.

Usage: Query only

[SENSe:]LIST:POWer[:SEQuence] <Frequency>, <RefLevel>, <RFAttenuation>, <ElAttenuation>, <FilterType>, <RBW>, <VBW>, <MeasTime>, <TriggerLevel>

This command configures and initiates the List Evaluation measurement.

The list can contain up to 200 entries (frequencies). You can define a different instrument setup for each frequency that is in the list.

If you synchronize the measurement with `*OPC`, the R&S FSW produces a service request when all frequencies have been measured and the number of individual measurements has been performed.

Note that using the command as a query initiates the measurement and returns the results if all frequencies have been measured. For more information on querying the results see `[SENSe:]LIST:POWer:RESult?`.

Parameters:

<Frequency> Defines the receive frequency. Each frequency corresponds to one list entry.

Range: 0 to Fmax
Default unit: Hz

<RefLevel> Defines the reference level for a list entry.

Range: -130 to 30
Increment: 0.01
Default unit: dBm

<RFAttenuation> Defines the RF attenuation for a list entry.

Range: 0 to 70
Increment: 1
Default unit: dB

<ElAttenuation> Defines the electronic attenuation for a list entry.

Range: 0 to 30
Increment: 1
Default unit: dB

<FilterType> Selects the filter type for a list entry. For more information see `[SENSe:]BANDwidth|BWIDth[:RESolution]:TYPE`.

<RBW> Defines the resolution bandwidth for a list entry.

<VBW> Defines the video bandwidth for a list entry.

<MeasTime> Defines the measurement time for a list entry.

Range: 1 μs to 16000 s

<TriggerLevel> The trigger level must be 0.

Return values:

<PowerLevel> Power level for each frequency included in the measurement.
The command returns up to 3 power levels for each frequency, depending on the number of evaluation modes you have turned on with `[SENSe:]LIST:POWer:SET`.
The result is a list of floating point values separated by commas.
The unit depends on `CALCulate<n>:UNIT:POWer`.

[SENSe:]LIST:POWer:SET <PeakPower>, <RMSPower>, <AVGPower>, <TriggerSource>, <TriggerSlope>, <TriggerOffset>, <GateLength>

This command defines global List Evaluation parameters.

These parameters are valid for every frequency you want to measure.

The state of the first three parameters (<PeakPower>, <RMSPower> and <AVGPower>) define the number of results for each frequency in the list.

Note that you have to set the trigger level *after* sending this command.

Parameters:

<PeakPower> ON | OFF

Turns peak power evaluation on and off.

*RST: ON

<RMSPower> ON | OFF

Turns RMS power evaluation on and off.

*RST: OFF

<AVGPower>	ON | OFF Turns average power evaluation on and off. *RST: OFF
<TriggerSource>	EXTernal | EXT2 | EXT3 | IMMediate | IFPower | RFPower | VIDeo Selects a trigger source. For more information see Configuring Triggered and Gated Measurements.
<TriggerSlope>	NEGative | POSitive Selects the trigger slop.
<TriggerOffset>	Defines the trigger delay. Range: negative measurement time to 30 *RST: 0 Default unit: s
<GateLength>	Defines the gate length for gated measurements. Setting 0 seconds turns gated measurements off. To perform gated measurements, the trigger source must be different from `IMMediate`. Range: 31.25 ns to 30 s *RST: 0 s

[SENSe:]LIST:POWer:STATe <State>

This command turns the List Evaluation off.

Parameters:

<State>	OFF *RST: OFF

10.3.13.2 Example: Performing List Evaluation

The following example shows a list evaluation with the following configuration.

No	Freq [MHz]	Ref Level [dBm]	RF Attenuation [dB]	El Attenuation [dB]	Filter	RBW	VBW	Meas Time	Trigger Level
1	935.2	0	10	---	Normal	1 MHz	3 MHz	440 µs	0
2	935.4	0	10	10	Channel	30 kHz	100 kHz	440 µs	0
3	935.6	0	10	20	Channel	30 kHz	100 kHz	440 µs	0

```
-----Measurement with synchronization via service request-----
*ESE 1
*SRE 32
```

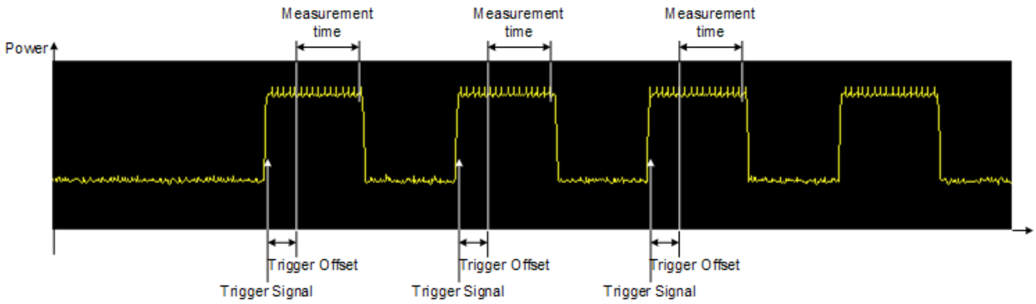
```
// Configures the status reporting system to produce a service request.
LIST:POW:STAT ON
//Turns on the list evaluation.
LIST:POW:SET ON,ON,OFF,EXT,POS,10us,434us
//Configures the global list evaluation settings and evaluates the peak and RMS power.
LIST:POW
935.2MHZ,0,10,OFF,NORM,1MHZ,3MHZ,440us,0,
935.4MHZ,0,10,10,CFIL,30KHZ,100KHZ,440us,0,
935.6MHZ,0,10,20,CFIL,30KHZ,100KHZ,440us,0;
*OPC
//Defines a list with 3 entries and initiates the measurement with synchronization to the end
//Analyzer produces a service request
//On service request:
SENS:LIST:POW:RES?
//Returns the results of the measurements, two for each frequency (peak and RMS power).

-----Initiliazing the measurement and querying results simultaneously-----
LIST:POW?
935.2MHZ,0,10,OFF,NORM,1MHZ,3MHZ,440us,0,
935.4MHZ,0,10,10,CFIL,30KHZ,100KHZ,440us,0,
935.6MHZ,0,10,20,CFIL,30KHZ,100KHZ,440us,0
//Defines a list with 3 entries, initiates the measurement and queries the results.
//Result example:
-28.3,-30.6,-38.1
```

10.3.14 Measuring the Pulse Power

All remote control commands specific to measuring the mean or peak pulse power (e.g. bursts in various telecommunications standards) are described here. This measurement is available via remote control only.

The Pulse Power measurement is a gated measurement that determines the power over a particular number of pulses. The measurement is controlled by an external trigger or the video signal. A separate trigger event is required for each burst included in the measurement. In case of an external trigger source, the trigger level corresponds to the TTL level. In case of a video signal, you can define any threshold.

The figure below shows the relations between the available trigger settings.



The measurement is always on trace 1, either with the peak detector to determine the peak power or the RMS detector to determine the RMS power. Overall, you can configure the measurement independent of the instrument setup with the commands listed below only, which results in faster measurements.

The Pulse Power measurement is incompatible to other measurement functions (e.g. marker functionality or statistics). If you use a command that controls those functions, the R&S FSW aborts the Pulse Power measurement.

The R&S FSW also aborts the Pulse Power measurement if you end the remote session.

The commands can be used in two different ways.

- Instrument setup, measurement and querying of the results in a single command line. With this method, there is the least delay between the measurement and the result output. However, it requires the control computer to wait for the response from the instrument.
- Instrument setup and querying of the result list at the end of the measurement: With this method, the control computer may be used for other activities while the measurement is being performed. However, more time is needed for synchronization via service request.

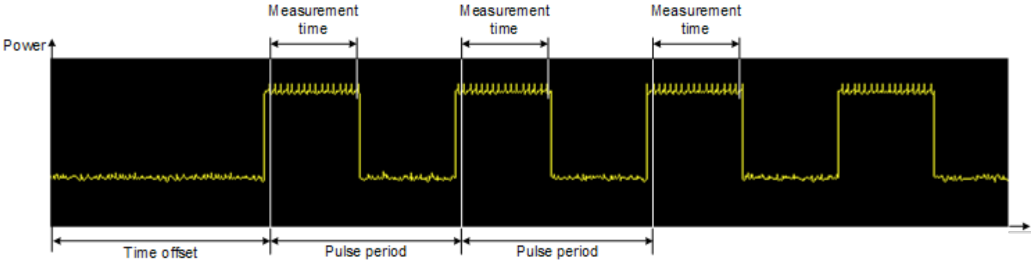
10.3.14.1 Performing Pulse Power Measurements

The following commands control pulse power measurements.

CALCulate<n>:MARKer<m>:FUNCtion:MSUMmary <TimeOffset>,<MeasTime>,<PulsePeriod>,<#OfPulses>

This command configures power measurements on pulses in the time domain.

To evaluate the pulse power, the R&S FSW uses the data captured during a previous measurement. The data recorded during the set measurement time is combined to a measured value for each pulse according to the detector specified and the indicated number of results is output as a list.



To get a valid result, you have to perform a complete measurement with synchronization to the end of the measurement before reading out the result. This is only possible for single sweeps. See also `INITiate:CONTinuous` on page 460.

Parameters:

<TimeOffset> Defines a time offset to start the measurement at the first pulse of a trace.
*RST: 0
Default unit: s

<MeasTime> Defines the measurement time.
Default unit: s

<PulsePeriod> Defines the pulse period

<#OfPulses> Defines the number of pulses to measure.

Example: `CALC:MARK:FUNC:MSUM 50US,450US,576.9US,8`
Evaluates data that contains 8 pulses during a measurement time of 450 µs and a pulse period of 576.9 µs. The evaluation starts with an offset of 50 µs.

[SENSe:]MPOWer:FTYPe <FilterType>

This command selects the filter type for pulse power measurements.

Parameters:

<FilterType> **CFILter**
NORMal
P5
RRC

[SENSe:]MPOWer:RESult[:LIST]?

This command queries the results of the pulse power measurement.

This command may be used to obtain measurement results in an asynchronous way, using the service request mechanism for synchronization to the end of the measurement.

If there are no results, the command returns an error.

Return values:

<PulsePower> List of pulse powers.
The number of values depends on the number of pulses you have been measuring.
The unit is dBm.

Usage: Query only

[SENSe:]MPOWer:[:SEQuence] <Frequency>, <RBW>, <MeasTime>, <TriggerSource>, <TriggerLevel>, <TriggerOffset>, <Detector>, <#OfPulses>

This command configures and initiates the pulse power measurement.

The R&S FSW caches all measurement parameters that you can set with this command. If you use the command repeatedly, the R&S FSW only changes those settings that you have actually changed before initiating the measurement. Thus, measurement times are kept as low as possible.

If you synchronize the measurement with `*OPC`, the R&S FSW produces a service request when all frequencies have been measured and the number of individual measurements has been performed.

Note that using the command as a query initiates the measurement and returns the results if all frequencies have been measured. For more information on querying the results see `[SENSe:]LIST:POWer:RESult?`.

Parameters:

<Frequency> Defines the pulse frequency.

Range: 0 to Fmax
Default unit: Hz

<RBW> Defines the resolution bandwidth.

<MeasTime> Defines the measurement time.

Range: 1 µs to 30 s

<TriggerSource> EXTernal | EXT2 | EXT3 | VIDeo

Selects a trigger source.
For more information see Configuring Triggered and Gated Measurements.

<TriggerLevel> Defines a trigger level.
The trigger level is available for the video trigger. In that case, the level is a percentage of the diagram height.
In case of an external trigger, the R&S FSW uses a fix TTL level.

Range: 0 to 100
Default unit: PCT

<TriggerOffset> Defines the trigger delay.

Range: 0 s to 30 s
*RST: 0 s
Default unit: s

<Detector> Selects the detector and therefore the way the measurement is evaluated.

MEAN
Calculates the RMS pulse power.

PEAK
Calculates the peak pulse power.

<#OfPulses> Defines the number of pulses included in the measurement.
Range: 1 to 32001

Return values:
<PowerLevel> Pulse power level.
The result is a list of floating point values separated by commas.
The unit is dBm.

[SENSe:]MPOWer:RESult:MIN?

This command queries the lowest pulse power that has been measured during a pulse power measurement.

If there are no results, the command returns an error.

Return values:
<PulsePower> Lowest power level of the pulse power measurement.
The unit is dBm.

Usage: Query only

10.3.14.2 Example: Performing a Pulse Power Measurement

The following example shows a pulse power measurement.

```
-----Measurement with synchronization via service request-----
*ESE 1
*SRE 32
// Configures the status reporting system to produce a service request.
MPOW:FTYP NORM
//Selects a Gaussian filter for the measurement.
MPOW 935.2MHZ,1MHZ,434us,VID,50,5us,MEAN,20;
*OPC
//Configures and initiates a measurement on 20 pulses with synchronization to the end.
//Analyzer produces a service request
//On service request:
MPOW:RES?
//Returns the results of the measurements (20 power levels).
MPOW:RES:MIN?
//Returns the lowest of the 20 power level that have been measured.

-----Initiliazing the measurement and querying results simultaneously-----
MPOW? 935.2MHZ,1MHZ,434us,VID,50,5us,MEAN,20
//Configures, initiates and queries the results of the measurement.
//Result example:
-105.225059509,-105.656074524,-105.423065186,-104.374649048,-103.059822083,-101.29511261,
-99.96534729,-99.7452468872,-99.6610794067,-100.327224731,-100.96686554,-101.450386047,
-102.150642395,-103.240142822,-105.95476532,-110.583129883,-115.7760849,-126.279388428,
-124.620399475,-116.97366333
```

10.4 Configuring the Result Display

The commands required to configure the screen display in a remote environment are described here.

The tasks for manual operation are described in chapter 6.1, "Result Display Configuration", on page 228.

10.4.1 General Window Commands

The following commands are required to configure general window layout, independant of the application.

Note that the suffix <n> always refers to the window *in the currently selected measurement channel* (see `INSTrument[:SELect]` on page 453).

DISPlay:FORMat <Format>

This command determines which tab is displayed.

Parameters:

<Format>

SPLit
Displays the MultiView tab with an overview of all active channels (See chapter 3.1, "R&S MultiView", on page 19).

SINGle
Displays the measurement channel that was previously focused.

*RST: SPL

Example: `DISP:FORM SING`

DISPlay[:WINDow<n>]:SIZE <Size>

This command maximizes the size of the selected result display window *temporarily*. To change the size of several windows on the screen permanently, use the `LAY:SPL` command (see `LAYout:SPLitter` on page 560).

Parameters:

<Size>	**LARGe** Maximizes the selected window to full screen. Other windows are still active in the background. **SMALl** Reduces the size of the selected window to its original size. If more than one measurement window was displayed originally, these are visible again. *RST: SMALl

Example: `DISP:WIND2:LARG`

DISPlay[:WINDow<n>]:SELect

This command sets the focus on the selected result display window.

This window is then the active window.

Example: `DISP:WIND1:SEL`
Sets the window 1 active.

Usage: Setting only

10.4.2 Working with Windows in the Display

The following commands are required to change the evaluation type and rearrange the screen layout for a measurement channel as you do using the SmartGrid in manual operation. Since the available evaluation types depend on the selected application, some parameters for the following commands also depend on the selected measurement channel.

Note that the suffix <n> always refers to the window *in the currently selected measurement channel* (see `INSTrument[:SELect]` on page 453).

LAYout:ADD[:WINDow]? <WindowName>,<Direction>,<WindowType>

This command adds a window to the display.

This command is always used as a query so that you immediately obtain the name of the new window as a result.

To replace an existing window, use the LAYout:REPLace[:WINDow] command.

Parameters:

<WindowName> String containing the name of the existing window the new window is inserted next to.
By default, the name of a window is the same as its index. To determine the name and index of all active windows, use the LAYout:CATalog[:WINDow]? query.

<Direction> LEFT | RIGHt | ABOVe | BELow
Direction the new window is added relative to the existing window.

<WindowType> text value
Type of result display (evaluation method) you want to add.
See the table below for available parameter values.

Return values:

<NewWindowName> When adding a new window, the command returns its name (by default the same as its number) as a result.

Example: `LAY:ADD? '1',LEFT,MTAB`
Result:
`'2'`
Adds a new window named '2' with a marker table to the left of window 1.

Usage: Query only

Manual control: See "Diagram" on page 229
See "Marker Table" on page 229
See "Marker Peak List" on page 229
See "Result Summary" on page 230
See "Spectrogram" on page 230

Table 10-2: <WindowType> parameter values for the Spectrum application

Parameter value	Window type
DIAGram	Diagram
MTABle	Marker table
PEAKlist	Marker peak list
RSUMmary	Result summary
SGRam	Spectrogram

LAYout:CATalog[:WINDow]?

This command queries the name and index of all active windows from top left to bottom right. The result is a comma-separated list of values for each window, with the syntax:

<WindowName_1>,<Index_1>..<WindowName_n>,<Index_n>

Return values:

<WindowName> string

Name of the window.
In the default state, the name of the window is its index.

<Index> **numeric value**
Index of the window.

Example: `LAY:CAT?`
Result:
`'2',2,'1',1`
Two windows are displayed, named '2' (at the top or left), and '1' (at the bottom or right).

Usage: Query only

LAYout:IDENtify[:WINDow]? <WindowName>

This command queries the **index** of a particular display window.

Note: to query the **name** of a particular window, use the `LAYout:WINDow<n>:IDENtify?` query.

Query parameters:

<WindowName> String containing the name of a window.

Return values:

<WindowIndex> Index number of the window.

Usage: Query only

LAYout:REMove[:WINDow] <WindowName>

This command removes a window from the display.

Parameters:

<WindowName> String containing the name of the window.
In the default state, the name of the window is its index.

Usage: Event

LAYout:REPLace[:WINDow] <WindowName>,<WindowType>

This command replaces the window type (for example from "Diagram" to "Result Summary") of an already existing window while keeping its position, index and window name.

To add a new window, use the `LAYout:ADD[:WINDow]?` command.

Parameters:

<WindowName> String containing the name of the existing window.
By default, the name of a window is the same as its index. To determine the name and index of all active windows, use the `LAYout:CATalog[:WINDow]?` query.

<WindowType> Type of result display you want to use in the existing window. See `LAYout:ADD[:WINDow]?` on page 557 for a list of available window types.

Example: `LAY:REPL:WIND '1',MTAB`
Replaces the result display in window 1 with a marker table.

LAYout:SPLitter <Index1>,<Index2>,<Position>

This command changes the position of a splitter and thus controls the size of the windows on each side of the splitter.

As opposed to the `DISPlay[:WINDow<n>]:SIZE` on page 556 command, the `LAYout:SPLitter` changes the size of all windows to either side of the splitter permanently, it does not just maximize a single window temporarily.

Note that windows must have a certain minimum size. If the position you define conflicts with the minimum size of any of the affected windows, the command will not work, but does not return an error.

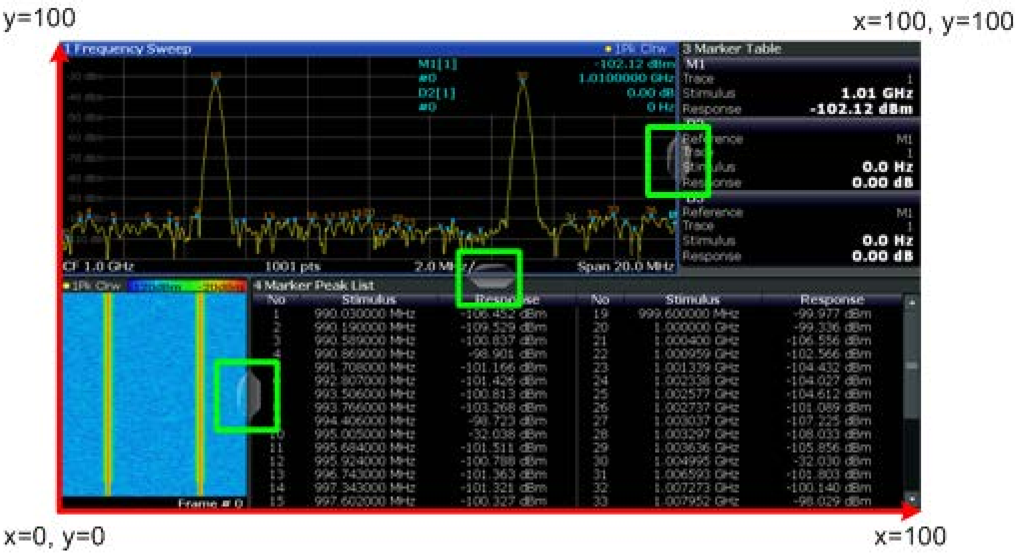


Fig. 10-1: SmartGrid coordinates for remote control of the splitters

Parameters:

<Index1> The index of one window the splitter controls.

<Index2> The index of a window on the other side of the splitter.