

# **LabXplorer**

**High Speed Multifunction PC  
Instrument**

User Manual

**DesignSoft**



# Contents

•	Introduction to Virtual Instruments .....	5
•	What is LabXplorer? .....	7
•	Specification .....	9
•	Safety Instructions .....	13
•	Technical Support .....	14
•	Panel Elements of LabXplorer .....	15
•	Accessories supplied .....	19
•	Installation .....	20
•	Quick Start .....	23
•	Digital Measurement .....	37
•	Device characteristic measurement .....	45
•	Test Card Experience .....	47
•	Generator Matching .....	52
•	Virtual Instruments in LabXplorer .....	53
•	Multimeter .....	53
•	Function Generator .....	55
•	Variable DC Supply .....	59
•	Oscilloscope .....	60
•	Signal/Spectrum Analyzer .....	64

- DC Parameter Analyzer.....68
- Digital Signal Generator.....71
- Logic Analyzer.....74
- Revision History.....79

# Introduction to Virtual Instruments

As measurement instruments become more and more sophisticated more and more software is needed to perform their advanced functions. In addition, in most of the modern instrument data are collected digitally, stored in memory and then evaluated by software. Obviously, the latter task can be ideally carried out by a personal computer where enormous software support can be provided as opposed to the limited programming capacity of a measurement instrument, unless you build in a personal computer (which also happens). Measurement results can be displayed on the PC monitor very similarly as on the screen of a traditional instrument. However, knobs and buttons are virtual and can be accessed by clicking on them with the mouse only.

Today we can see an interesting convergence of computers and measurement instruments.

The most advanced instruments often include a board like a personal computer sometimes even a PC operating system like Windows is included. Even though there are still real buttons and knobs on the instrument but you can access everything by simple click of the mouse, which is also part of the system. The other direction is just the opposite. You can extend your PC with a data acquisition and generator hardware, either an internal extension card or an external unit connected to the PC. Most operations on the collected data are done by the PC, controlling the instrument is carried out by Windows style buttons and fields.

LabXplorer represents the latter direction. It is an external unit connected to the PC through USB. It includes a powerful data acquisition unit and digital and analog signal generators.

To accelerate the operation the LabXplorer also includes a DSP chip with various programs which can be uploaded from the PC.

If you use LabXplorer together with TINA circuit simulation software there is an interesting concept to mention. In TINA you can also display simulation results on virtual instruments. This has the advantage that you can investigate your design like in the practice with an instrument. Now the virtual instruments in TINA and LabXplorer are exactly the same. Therefore, once you have designed a circuit with simulation it is extremely easy to measure it and bring to life.

# What is LabXplorer?

With LabXplorer you can turn your laptop, desktop, or network server machine into a powerful, multifunction test and measurement instrument. The LabXplorer is the extended version of TINALab II with lots of additional and remote measurement capability. Like with TINALab II, whichever instrument you need — multimeter, oscilloscope, spectrum analyzer, logic analyzer, parameter analyzer, arbitrary waveform generator, or digital signal generator - it is at your fingertips with a click of the mouse. In addition, LabXplorer can be used with the TINA circuit simulation program in desktop environment for comparison of simulation and measurements as a unique tool for circuit development, troubleshooting, and the study of analog and digital electronics. In network environment, together with the TINACloud software running in a browser, the clients can reach collected measured data transforming the workstations to powerful instrument.

LabXplorer includes a DC to 50MHz bandwidth, 10/12 bit resolution dual-channel **Digital Storage Oscilloscope**. Due to its advanced equivalent-time sampling technology, LabXplorer can acquire any repetitive signal with up to **4 GS/s equivalent sampling rate**, while in single shot mode the sampling rate is 20 MS/s. The full scale input range is  $\pm 80\text{V}$ , with 5mV to 20V/div ranges.

The synthesized **Function Generator** provides sine, square, ramp, triangle and arbitrary waveforms from DC to 4MHz, with logarithmic and linear sweep, and modulation up to 5V peak to peak on  $50\Omega$ . Arbitrary waveforms can be programmed via the high level, easy to use language of TINA's Interpreter.

Working automatically in conjunction with the Function Generator, the **Signal Analyzer** measures and displays Bode amplitude and phase diagrams, Nyquist diagrams, and also works as a **Spectrum Analyzer**.

Digital I/O for the high-tech **Digital Signal Generator** and **Logic Analyzer** instruments allow fast 16-channel digital testing up to 40MHz.

The optional **Multimeter and Impedance/Semiconductor Parameter Analyzer** for LabXplorer allows DC measurements in ranges from 10mV to 100V and from 10mVrms to 20Vrms in AC. Current ranges are from 1mA to 1A. It can also measure DC resistance in ranges from  $1\Omega$  to  $10\text{M}\Omega$ .

To power circuits for experiments or demonstration LabXplorer offers two fixed (+5V, -5V) and two variable DC supply. The **Variable DC Power Supplies** works in range from 0 to 15V and 0 to -15V.

Using LabXplorer with **TINA or TINACloud**, DesignSoft's popular circuit simulation programs, give you the unique capability to have circuit simulation and real time measurements in the same integrated environment. Those provide invaluable tools for troubleshooting and bring your designs to life by comparing simulated and measured results. You can also plug experimenter cards into the slot on the front of LabXplorer, allowing you to simulate, measure, and troubleshoot virtually the whole range of analog, digital and programmable electronics.



# Specification

## SYNTHESIZED FUNCTION GENERATOR

WAVEFORM OUTPUT	
Amplitude	$\pm 10\text{V}$ , $\pm 5\text{V}$ (into 50 Ohm)
Nominal Impedance	50 Ohm
Protection	Short circuit protection in range of $\pm 15\text{V}$
Waveform	Square, Ramp, Triangle, DC, Arbitrary
Bandwidth	50MHz
Frequency range	
Sine	4MHz
Square, Triangle	500kHz
Arbitrary	depends on waveform
Synch Output (indicates 0V crossing)	5V CMOS level

D/A CONVERTER	
Resolution	10 [12] bits ( $<0.1\%$ of full scale), monotonic
Update Rate	40 MSPS

WAVEFORM MEMORY	
Type	Fast static RAM
Capacity	Min. 65,536 * 16bits

## DIGITAL OUTPUTS

# of Lines	16
Connection	all 16 bits wired to a shrouded male connector 8 least significant bits wired to 2mm sockets
Compatibility	5V tolerant devices (5V TTL/CMOS, 3.3V LVCMOS)
Protection	Short circuit protection in range of $\pm 15\text{V}$

## ANALOG INPUTS

# of Channels	2
Input Configurations	Single ended
Full Scale Input Range	$\pm 80V$
Nominal Impedance	1 MOhm / 25pF

A/D CONVERSION	
Resolution	10 [12] bit (no missing codes)
Sample Rate (non-repetitive signal)	20 MSPS/channel
Equivalent Sample Rate (repetitive signal)	4 GSPS/channel

SYSTEM CHARRACTERISTIC	
Bandwidth (-3dB)	DC .. 50MHz DC-coupled 2Hz .. 50MHz AC-coupled

FRAME MEMORY / CHANNEL	
Type	Fast static RAM
Capacity	min. 65536 * 16 bit
Input mode	Digitized analog and/or digital

VERTICAL MEASUREMENT RANGES	
Scale Factor	5mV/div .. 20 V/div
Attenuation Factors	x1, x2, x5

HORIZONTAL MEASURE RANGES	
Scale Factor	10 s/div .. 2 $\mu$ s/div
Scale Factor (repetitive signal)	10 s/div .. 10ns/div
Attenuation Factors	x1, x2, x5

TRIGGER SOURCE	
Analog	Channels A, B, external
Digital	16 * 8 bits trigger pattern

TRIGGER RESPONSE (WAVEFORM GENERATING)	
Free Run	[Internal] trigger then go
Single	One waveform then stop
Continuous	One waveform then wait for a trigger

## DIGITAL INPUT

# of Lines	16
Connection	all 16 bits wired to a shrouded male connector 8 least significant bits wired to 2mm sockets
Compatibility	3.3V CMOS/TTL, 5V CMOS/TTL
Overvoltage Protection	min. $\pm 15V$

## MULTIMETER/DC PARAMETER ANALYZER (OPTIONAL)

A/D CONVERSION	
Resolution	16 bit

AC VOLTAGE	
Measurement Range	10mV..20Vrms
Range Factors	x1, x2, x5
Input Impedance	10 MOhm
Bandwidth	10Hz..50kHz

DC VOLTAGE	
Measurement Range	10mV..50V
Range Factors	x1, x2, x5
Input Impedance	10 MOhm

AC CURRENT	
Measurement Range	1mA..1Arms
Range Factors	x1, x2, x5

DC CURRENT	
Measurement Range	1mA..1A
Range Factors	x1, x2, x5

RESISTANCE/IMPEDANCE	
Measurement Range	10Ohm..1MOhm
Range Factors	x1, x2, x5

SEMICONDUCTOR PARAMETER ANALYZER	
Full Scale DC Output Voltage on B	$\pm 10V$
Resolution	1.22mV
Full Scale DC Output Current on B	$\pm 2mA$
Resolution	244nA
Full Scale Input Current Range into D-	$\pm 50mA$

## INTERFACE

USB 1.1	full speed
---------	------------

## DC POWER SUPPLY

Fixed DC output	
+5V	0.4A
-5V	0.1A
Variable DC output	
+15V	0.1A
-15V	0.2A

## EXTERNAL POWER SUPPLY

Type	switching, external
Input	100..240VAC, 47..63Hz
Output	
XP Power AEH45UM32	+5V/5A, +15V/2A, -15V/0.8A

## GENERAL

Weight	ca. 1kg
Color	Silk white
Dimensions (W x H x D)	291 x 75 x 199 mm

# Safety Instructions

Read these safety instructions before you start using LabXplorer.

1. LabXplorer includes an external power supply with a three conductor power cord with protective earthing conductor and a plug with earthing contact. The cord connecting the power supply and LabXplorer includes the earthing conductor, so LabXplorer is also earthed. Always plug the power cord into a proper grounded outlet before connections are made to any measuring circuits. **Use the Proper Power Cord.** Use only the power cord specified for this product and certified for the country of use.
2. If LabXplorer is not earthed, all accessible metal parts might be hazardous and cause an electrical shock.
3. Be cautious when you measure voltages higher than 25V AC or 60 V DC. High voltages are dangerous. For safety reasons, you must NEVER connect LabXplorer to the mains supply.

# Technical support

To troubleshoot and fix hardware or software issues contact us via the preferred communication channel.

Frequently Asked Questions

<http://www.tina.com/English/tina/FAQ2&id=Tina>

Questions and Answers

<http://www.tina.com/English/tina/tssearch>

Discussion Forum

<http://forum.designsoftware.com/>

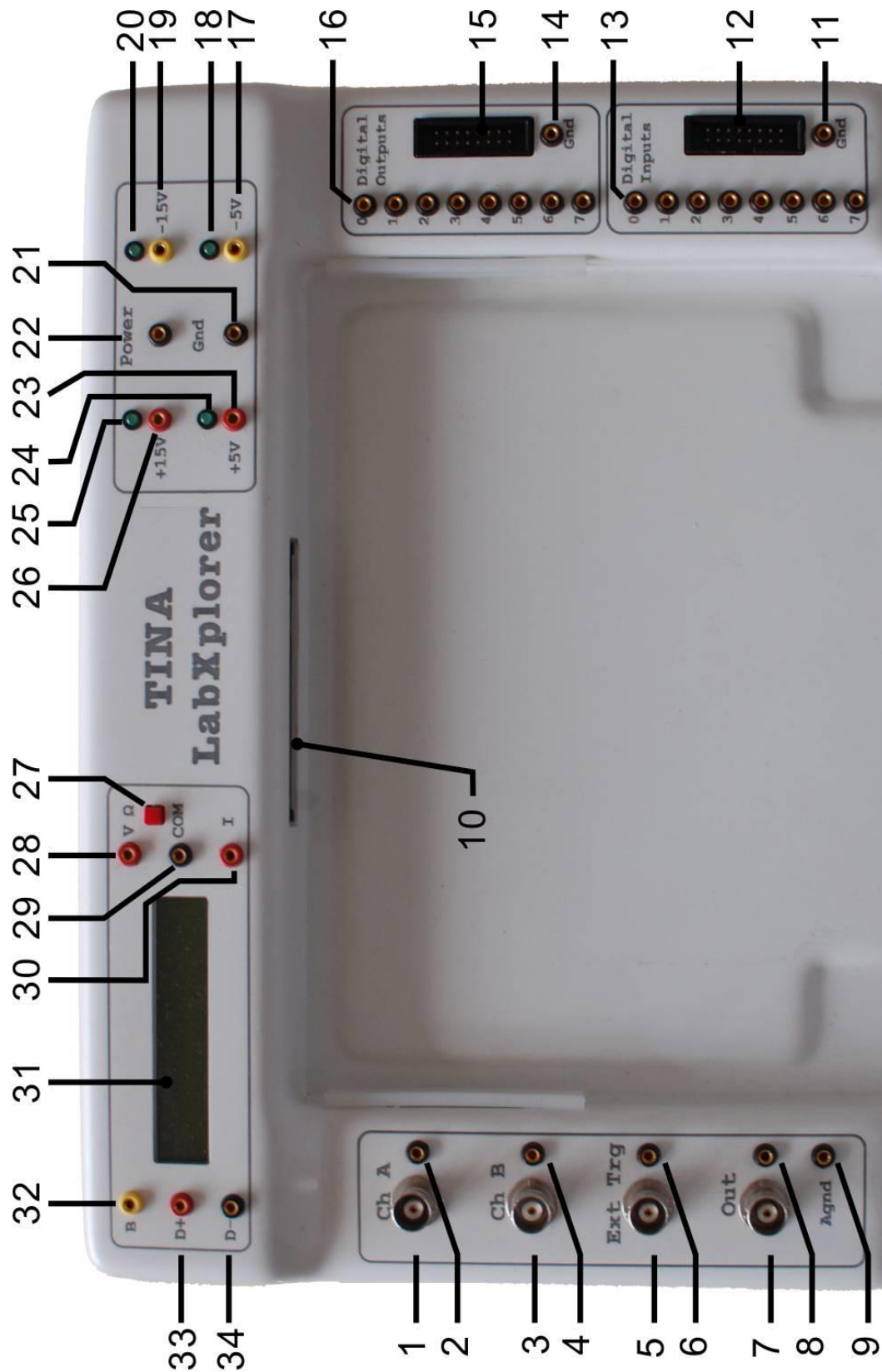
Technical Support Page

<http://www.tina.com/English/tina/techsup&id=Tina>

Video tutorials

<http://www.tina.com/English/tina/techsup&id=Tina>

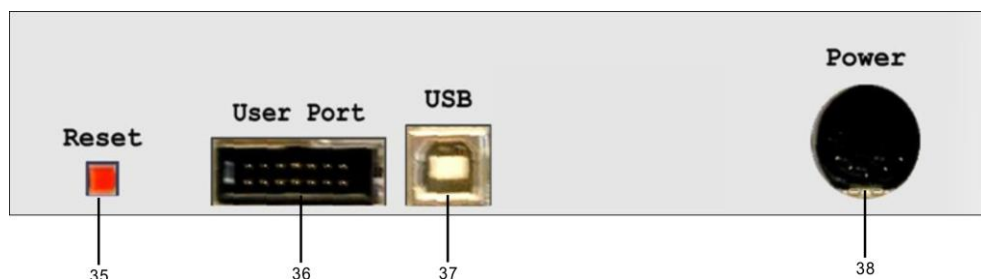
# Panel elements of LabXplorer



1. Ch A Input (BNC connector) This is the signal input for channel A. The outer (ground) connection is galvanically connected to the instrument ground and consequently to the safety earth contact of the line/mains plug.
2. Ch A Input (2mm socket connector) This is the signal input for channel A.
3. Ch B Input (BNC connector) Signal input for channel B. The outer (ground) connection is galvanically connected to the instrument ground and consequently to the safety earth contact of the line/mains plug.
4. Ch B Input (2mm socket connector) Signal input for channel B.
5. Ext Trg Input (BNC connector) External trigger signal input if external triggering is selected. The outer (ground) connection is galvanically connected to the instrument ground and consequently to the safety earth contact of the line/mains plug.
6. Ext Trg Input (2mm socket connector) External trigger signal input, if external triggering is selected.
7. 50 $\Omega$  Output (BNC connector) Short-circuit-protected signal output of the function generator. The output impedance is 50 $\Omega$ , and the maximum output amplitude is 20Vpp (open circuit) or 10Vpp when terminated with 50  $\Omega$ .
8. 50 $\Omega$  Output (2mm socket connector) Short-circuit-protected signal output of the function generator.
9. Analog ground
10. Experimenter Board (80 pins connector) Slot for experimenter boards. Experimenter boards are ready-to-use circuits for measurement experiments manufactured by DesignSoft, Inc. For details, see [www.tina.com](http://www.tina.com) LabXplorer experimenter boards.
11. Digital ground
12. Digital Inputs (shrouded male connector) Signal inputs of the Logical Analyzer. The inputs are protected against external DC voltages up to max.  $\pm 15\text{V}$ .
13. Digital Inputs (2mm socket connector) Signal inputs of the Logical Analyzer. The inputs are protected against external DC voltages up to max.  $\pm 15\text{V}$ .
14. Digital ground
15. Digital Outputs (2mm socket connector) Signal outputs of the Digital Signal Generator. The outputs are protected against external DC voltages up to max.  $\pm 15\text{V}$ .
16. Digital Outputs (shrouded male connector) Signal outputs of the Digital Signal Generator. The outputs are protected against external DC voltages up to max.  $\pm 15\text{V}$ .
17. -5V fixed power supply output
18. -5V power on LED



19. -15V programmable power supply output. Voltage can be changed from 0 to -15V.
20. -15V power on LED
21. Digital ground
22. Digital ground
23. +5V fixed power supply output
24. +5V power on LED
25. +15V power on LED
26. +15V programmable power supply output. Voltage can be changed from 0 to 15V.
27. Multimeter mode pushbutton
28. V  $\Omega$  (2mm socket connector) Input terminal to measure voltage and resistance with the multimeter. **Attention! This input does not work unless a LabXplorer Multimeter Card is installed.**
29. COM (2mm socket connector) Common ground of the multimeter measuring V,  $\Omega$ , I. This ground is independent of all the other ground connections of the instrument (Gnd, Agnd).
30. I (2mm socket connector) Input terminal to measure current with the multimeter. **Attention! This input does not work unless a LabXplorer Multimeter Card is installed.**
31. LCD display. If the LabXplorer Multimeter Card installed and the instrument is unconnected state the display shows measured values. Default mode is AC voltage autorange. Mode can be change by mode button (27).  
If the instrument is communicating through the USB to the PC software display shows information messages. The measured values of the multimeter
32. B (2mm socket connector) Output terminal to the base (or gate) pin of the transistor for measuring its transfer characteristic.
33. D+ (2mm socket connector) Output terminal to the DUT (device under test) providing programmable voltage for measuring two-terminal device characteristics.
34. D- (2mm socket connector) Input terminal to measure the DUT current.



35. Reset button
36. User Port (shrouded male connector) Reserved for further development.

37. USB interface **Attention! All terminals of the USB interface are galvanically connected with the instrument and subsequently with protective (safety) earth potential.**
38. External power source connector

# Accessories supplied

1. TINALab Basic install CD (LabXplorer virtual instrument set)
2. LabXplorer User Manual
3. External Power Supply (Fortron/Source, Model No: FSA30P32-S or XP Power AEH45UM32)
4. AC Power Cord (optional)
5. USB A-B Cable, 1.8m
6. 2 Oscilloscope Probe Kits (K&H, LF-101)
7. BNC Test Lead (K&H, LF-56)
8. Breadboard (K&H, RH-21)
9. Test Leads, 15cm, Standard color (K&H, KLG-0.65-2)
10. Test Leads, 30cm, Standard color (K&H, KLG-0.65-2)
11. Small wire set, 14 length/each 5pcs (K&H, SWS-03)
12. Test kit

200Ohm, 0.6W, Metal film, 1%
10nF, 400V, Foil capacitor, 10%
8.2mH, 0.1A, Inductor coil, 10%

Note: Subject to change without notice

# Installation

You need Windows 7 or higher system to operate LabXplorer and note, first install the software then connect the instrument to the computer.

LabXplorer operates completely through virtual instruments, so you must install its operating software on the PC. It is necessary to install the software provided on the CD included with LabXplorer so that your PC will have the necessary drivers to access LabXplorer through a set of virtual instruments.

Note that once the drivers are installed, you can use the same instruments from within TINA software. As an alternative installation method, you could instead install the drivers together with TINA using the Custom Installation option and selecting TINALab II & LabXplorer Device Driver.

To begin the installation, simply insert the CD into your CD-ROM drive. The setup program will automatically start if the Auto-Run function has been enabled. If not, start setup.exe from your CD. Follow the steps of the setup procedure standard with most Windows program.

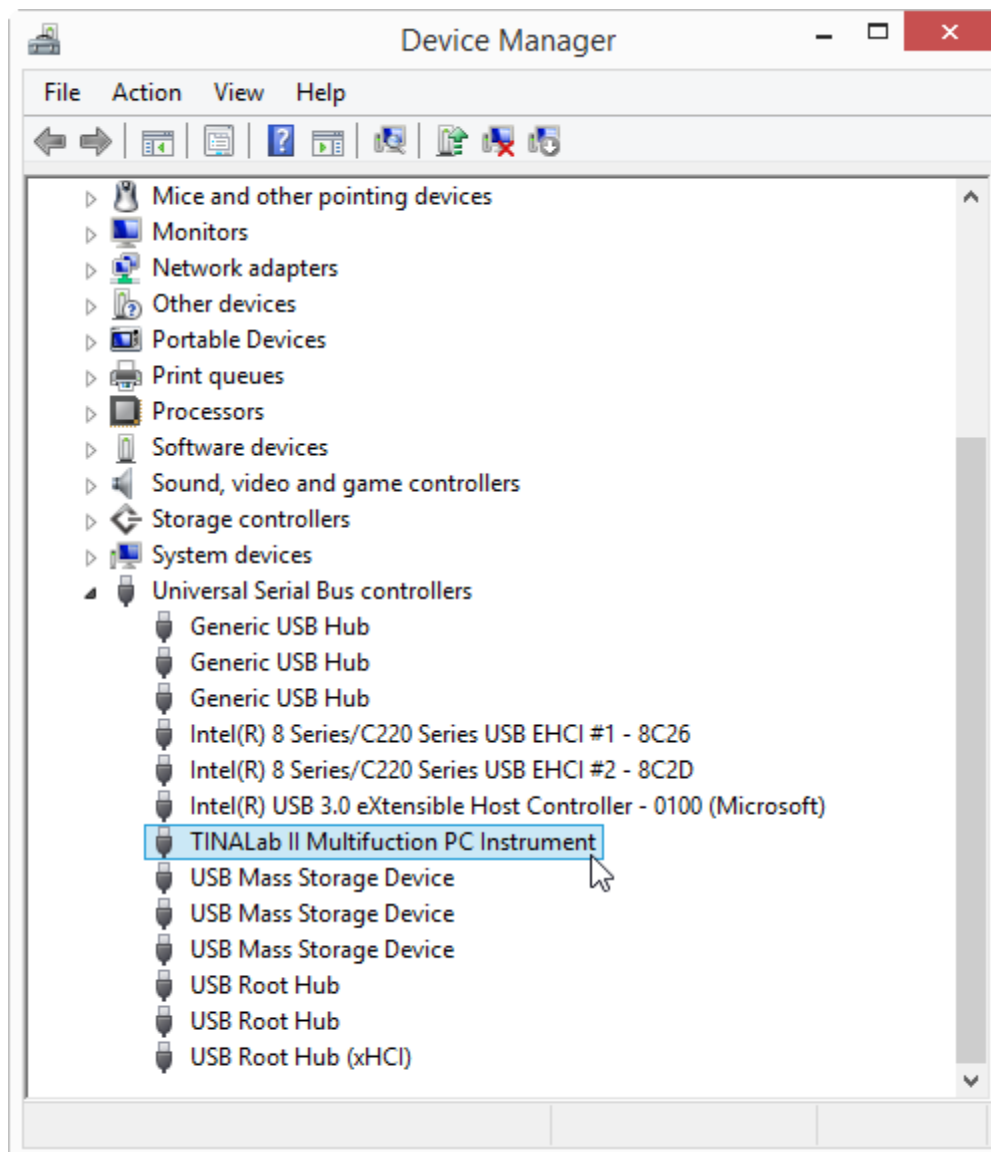
When prompted by the Windows Security ‘Would you like to install this device software?’, click Install.

Now, connect LabXplorer to its power supply, and then plug the power supply into a mains outlet provided with a protective earth.

Use the USB cable provided to connect LabXplorer to the personal computer.

The New Hardware wizard window should appear.

When the install process is over, you can easily check the result. Start the Device Manager. Choose Universal Serial Bus controllers in the Information Tree. The ‘TINALab II Multifunction PC instrument’ refers to LabXplorer.



When the installation is finished, select TINALab Basic. The instrument will be recognized automatically.



If the instrument is switched off or is not able to communicate then the following window will appear.



Fix the issue then press OK. Now you can use any of the virtual instruments in the window.

The following chapter presents a quick brief introduction, while in later chapters a detailed description is provided for each instrument.

# Quick Start

This chapter introduces the use of LabXplorer through examples. For brevity, we will explain only those controls which are needed for these examples. The detailed description of all instruments is given in later chapters.

## Analog Measurements

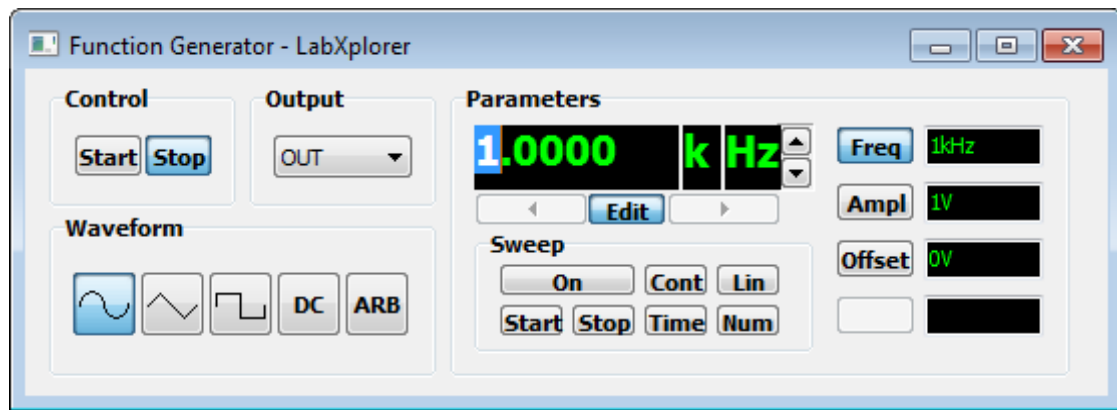
Let us illustrate the analog measurement capabilities of LabXplorer using the Signal Generator, Multimeter and Oscilloscope.

After installation, let's start LabXplorer from the Start menu of Windows. The following window will appear.

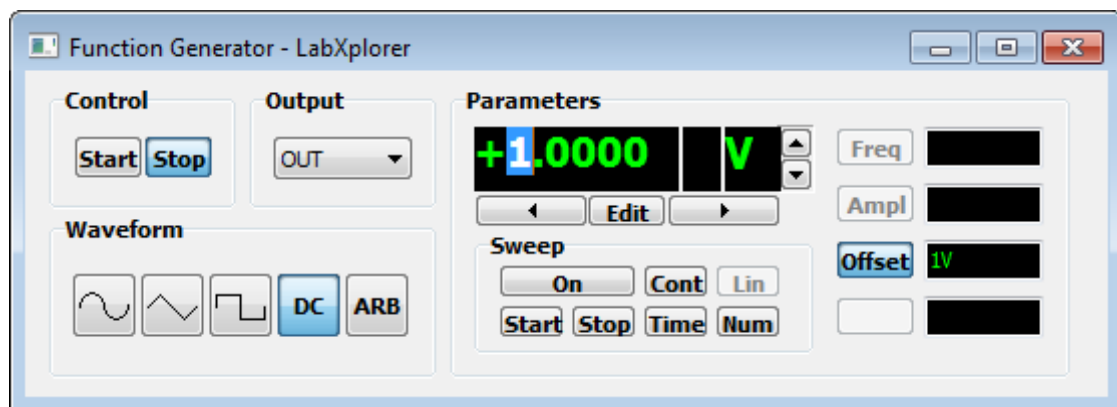


On LabXplorer, let's connect Out to Ch A using the cable provided with the instrument. (Out is the output of the Function Generator, Ch A is the A input of the Oscilloscope.)

Click the Function Generator button. The Function Generator will appear.

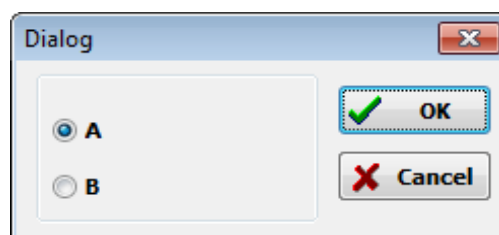


As you can see, the default settings are for a 1kHz sine wave at 1V. Now, choose DC, set 1V, then press the Start button.



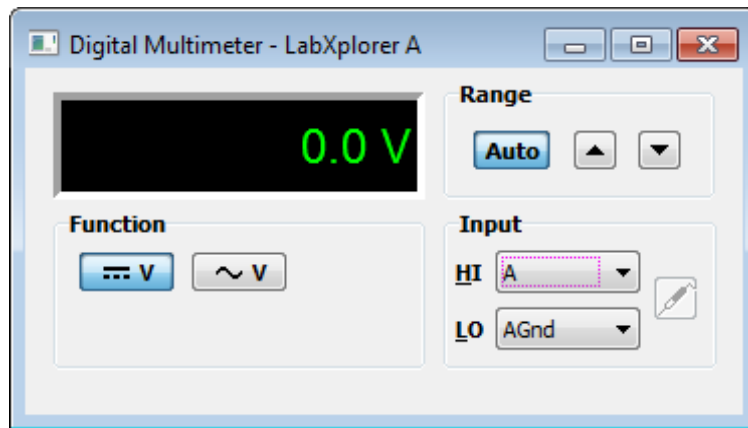
Now invoke the Multimeter by clicking the multimeter button in the LabXplorer window.



If you do not have the optional Multimeter Extension Card (MMC) installed in LabXplorer, the following window will appear. Here you can choose one of the analog channels as input.







Choose A and click OK. The following window will appear.





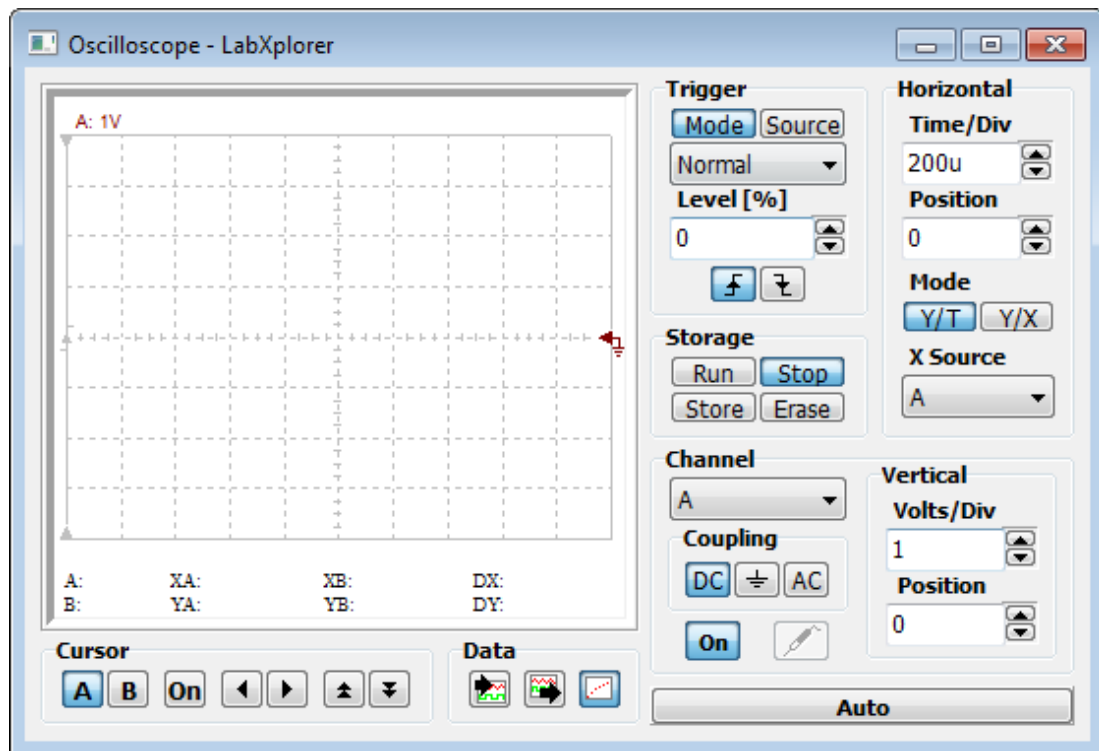
Now, click the  button. The instrument should measure the set 1 V DC voltage delivered by the signal generator. Change the voltage on the Signal Generator by clicking the  buttons and check the result on the Multimeter. You can set any value by clicking the Edit button, typing in a value, and pressing Enter.

Now click the  button on the Signal Generator. The amplitude will go back to 1V, indicating that the instrument is generating a sinusoidal voltage with a peak amplitude of 1V. By default, the frequency will be 1kHz, but you can change this by clicking the  button and setting the value with the  buttons or by typing a value in after pressing the Edit button.

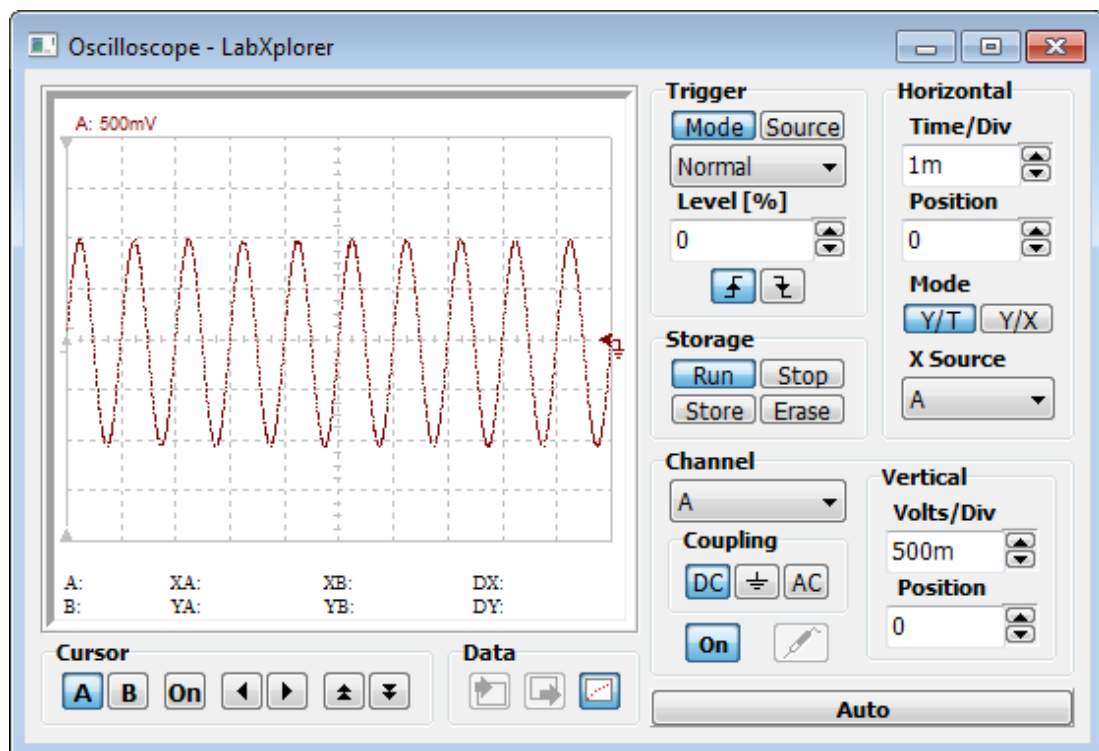
Now click the  button on the Multimeter. With the (peak) amplitude of the signal on the Signal Generator set to 1V, the Multimeter should show approximately 707 mV, the effective value of a 1V peak sine wave.

You should only use the other buttons of the Multimeter if your LabXplorer has the MMC installed. Furthermore, these functions are available only via the V, COM, and I connectors of LabXplorer.

Now let us investigate the sinusoidal signal using the Oscilloscope. Click the Oscilloscope button on the TINA Lab Basic panel, the following window will appear.






Make sure that the frequency of the Signal Generator is 1kHz. On the Oscilloscope set Channel to A, set the Horizontal Time/Div field to 1m, and press the Run button. The following window will appear:



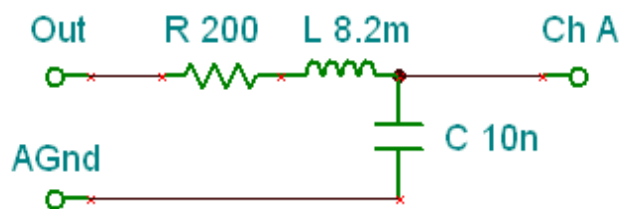
Play with the Time/Div and Volts/Div controls to see their effects on the displayed curve.

The displayed curve will appear to be “synchronized” using triggering. Triggering forces the starting point of the repetitive drawing of a signal to always be at the same phase of the signal.

The sine wave at the left side of the instrument starts at the trigger level. You can also determine whether the drawing is triggered at the rising or falling part of the signal. You can select between the two triggering modes using the  buttons.

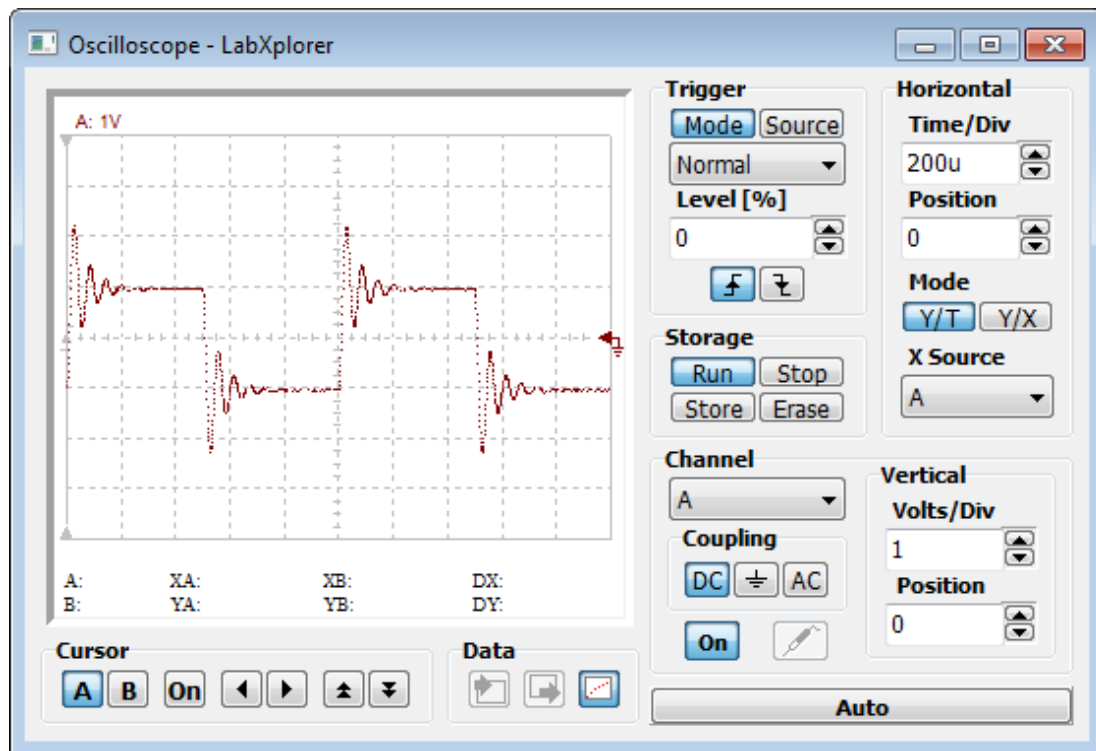
Change the wave form on the Signal Generator by clicking the  and  buttons and see the waveform on the Oscilloscope immediately.

To further acquaint ourselves with LabXplorer and confirm its correct operation, let's see a simple RLC circuit shown in the figure below. (The components are enclosed.)



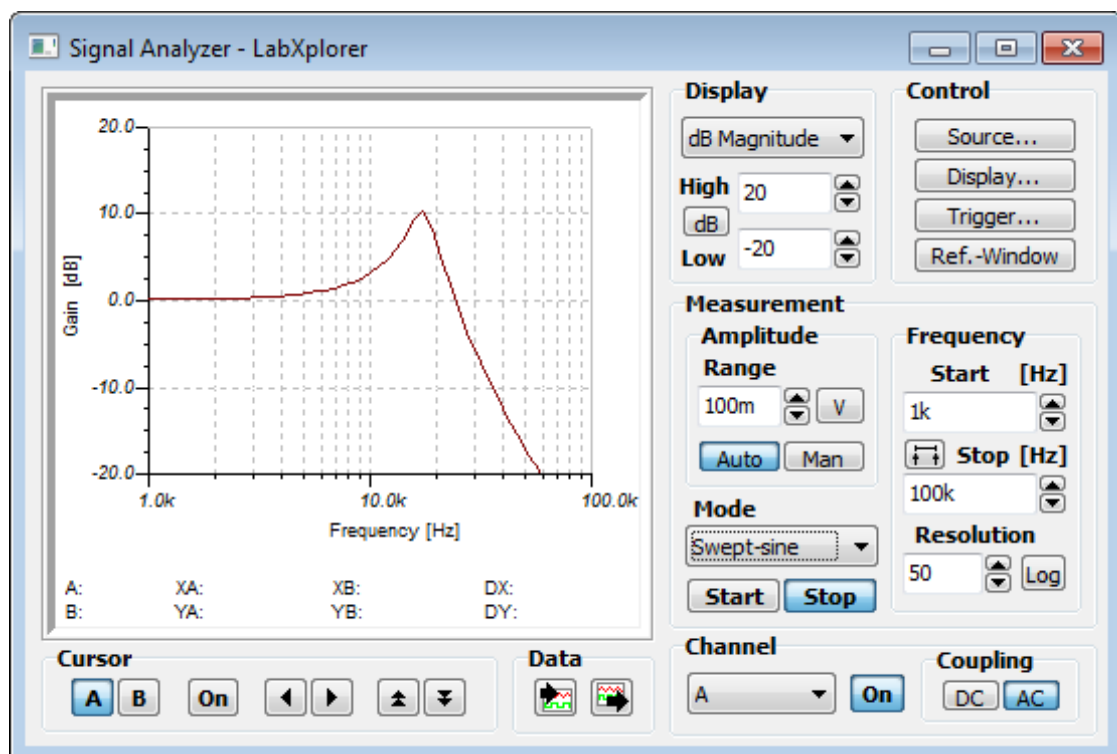
Now, let's assemble the RLC circuit. You should connect the output of the Function Generator (Out) to the circuit input, while the output of the circuit should be connected to either channel of the Oscilloscope (here we use Ch A). And finally, do not forget to tie analog ground to AGnd.

For the first trial, click the Function Generator and set frequency to 1kHz with peak amplitude of 1V. Choose square waveform and press the Start button. Click the Oscilloscope, then click Run and then Auto. TINA will initialize the measurement devices. See the following figure for the results. Try different input waveforms using the Function Generator.



Now, click the Signal Analyzer button, set Start frequency to 1 kHz, Stop frequency to 100 kHz, Resolution to 50, confirm that the A Channel is selected, and then press Start.

The Signal Analyzer will automatically change the amplitude range during the sweep to achieve maximum sensitivity at the input.



For frequency domain analysis, we use the **Spectrum Analyzer**, both for periodic and non-periodic signals.

Let's see the Fourier spectrum of the circuit response just viewed with the Oscilloscope.

Make the following settings:

- Press the Ref. Window button to invoke the window where the reference level and window function can be selected.

The reference level is used as a reference for logarithmic representation of the results, while the window function is for reducing spectrum leakage problems due to the finite time length records. Depending on the particular window function, you will establish a compromise between amplitude accuracy and frequency resolution. Select Hanning for good frequency resolution with somewhat reduced amplitude accuracy, or select Flattop for improved amplitude accuracy at the expense of somewhat poorer frequency resolution (compared to the Hanning window).

- Select the Flattop window function.
- Press the Display button.
- 

## Display

Select Lin. Magnitude

Set **High** to 2

Set **Low** to 0

## Mode

Select Ampl. Spectrum

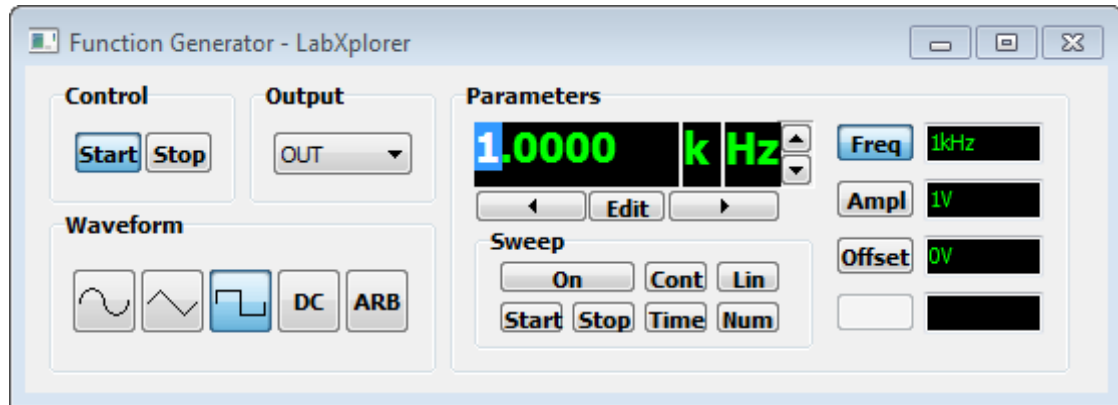
## Frequency


Set **Start** to 100

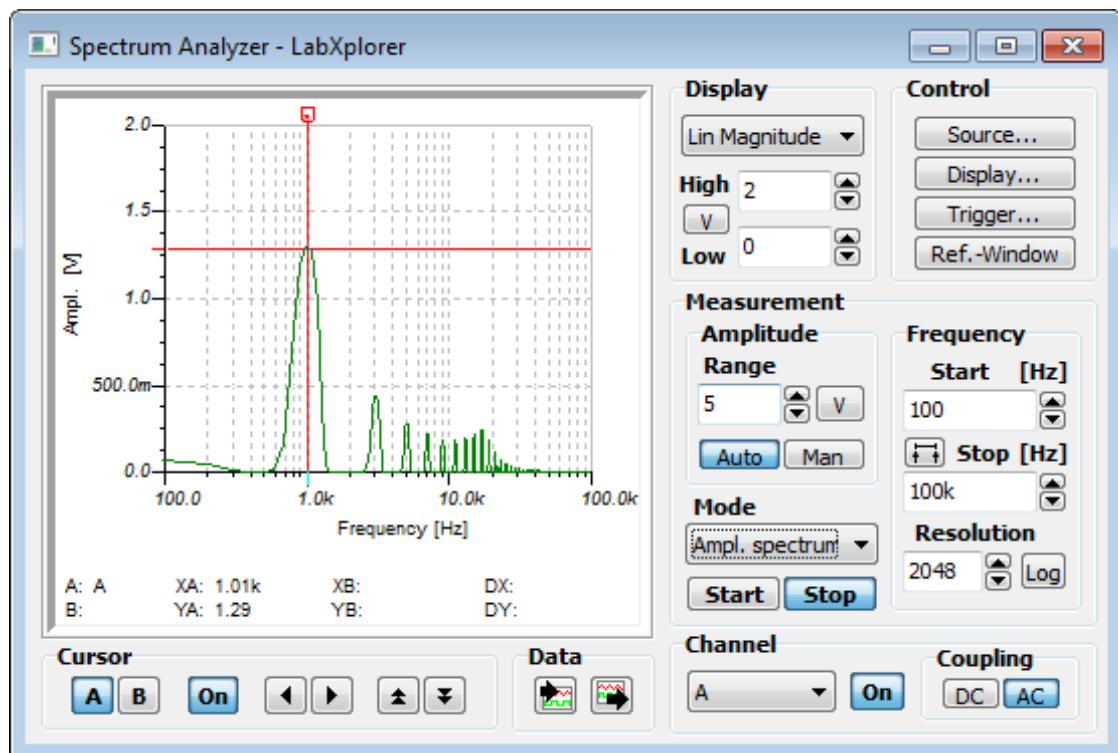
Set **Stop** to 100k

- Invoke the Function Generator, make sure it is set to a 1V, 1kHz square wave, and press its **Start** button. If the Function Generator is not on

screen, the simplest way to invoke it is to press the Source button on the Signal Analyzer.



- Now press the **Start** button on the Signal Analyzer. The Fourier Spectrum will appear.
- Press the On button under **Cursor** and with the  buttons position the A cursor to the leftmost largest spectral line at 1kHz.

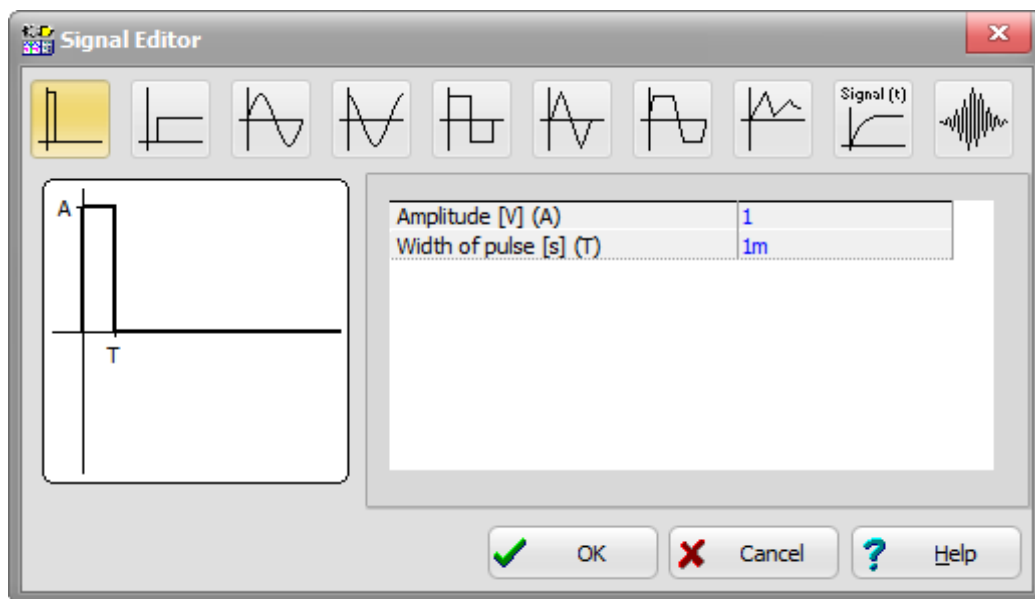


Observe the  $Y_A=1.28V$  value at the bottom of the instrument. This is very close to the well-known  $4/\pi=1.2733$  value of the fundamental Fourier component of our 1kHz, 1V square wave excitation. Looking at the Bode diagram already measured, you can check that at 1kHz the circuit has no

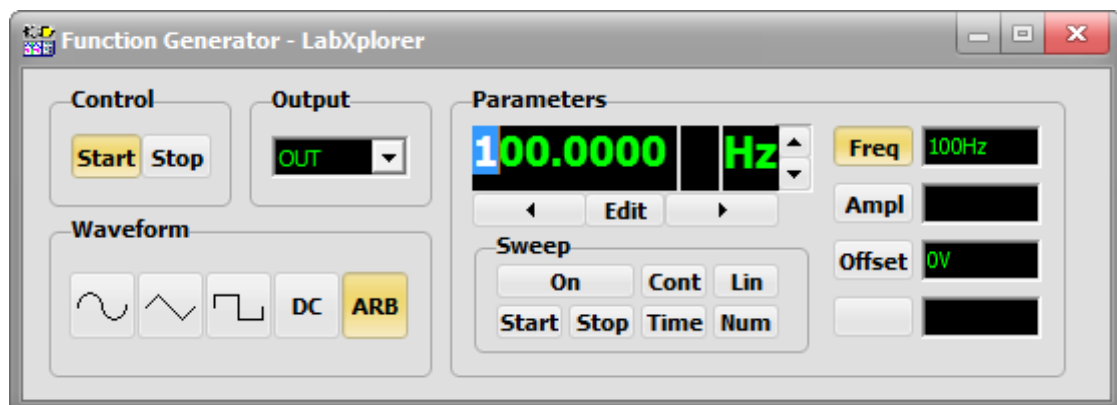
attenuation. The small maximum around 18kHz is due to the damped sinusoidal response of the circuit.

Now let's examine the spectrum of a non-periodic signal. Analyze the response of our circuit to a single pulse of 1V peak and 1ms width.

- Press the **Source** button on the Signal Analyzer to invoke the Function Generator.
- Press the **Arb** button to see more waveforms. Press the first button (Pulse), set the Pulse width to 1m, and press OK.



Observe that after pressing OK the frequency of the Function generator will be changed, in our case to 100Hz. Do not change this frequency unless you want to transform the length of the pulse. The pulse width is proportional to the generator frequency.



- Press the **Ref. Window** button in the Signal Analyzer and select the Uniform window function.
- Press the **Trigger** button and (under Trigger/Mode) select Single to establish that the signal analyzer will run only once, triggered by the single pulse.
- Press the **Display** button.

## Display

Select Lin. Magnitude

Set **High** to 2m

Set **Low** to 0

## Mode

Select Ampl. Spectrum Density

## Frequency

Set **Start** to 100

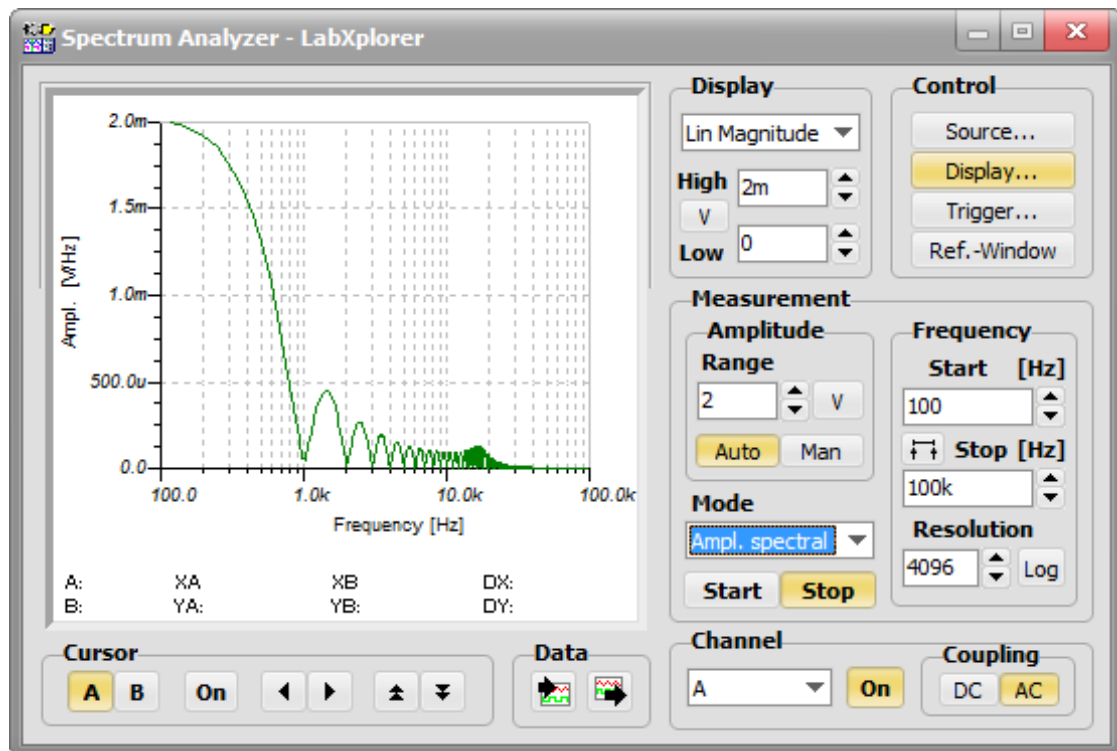
Set **Stop** to 100k

## Resolution

Set 4096


- Press the **Start** button. The Signal Analyzer will go into a wait mode.
- Now press the **Start** button on the Function generator. The Fourier spectrum will appear on the Signal Analyzer and it will **Stop**.

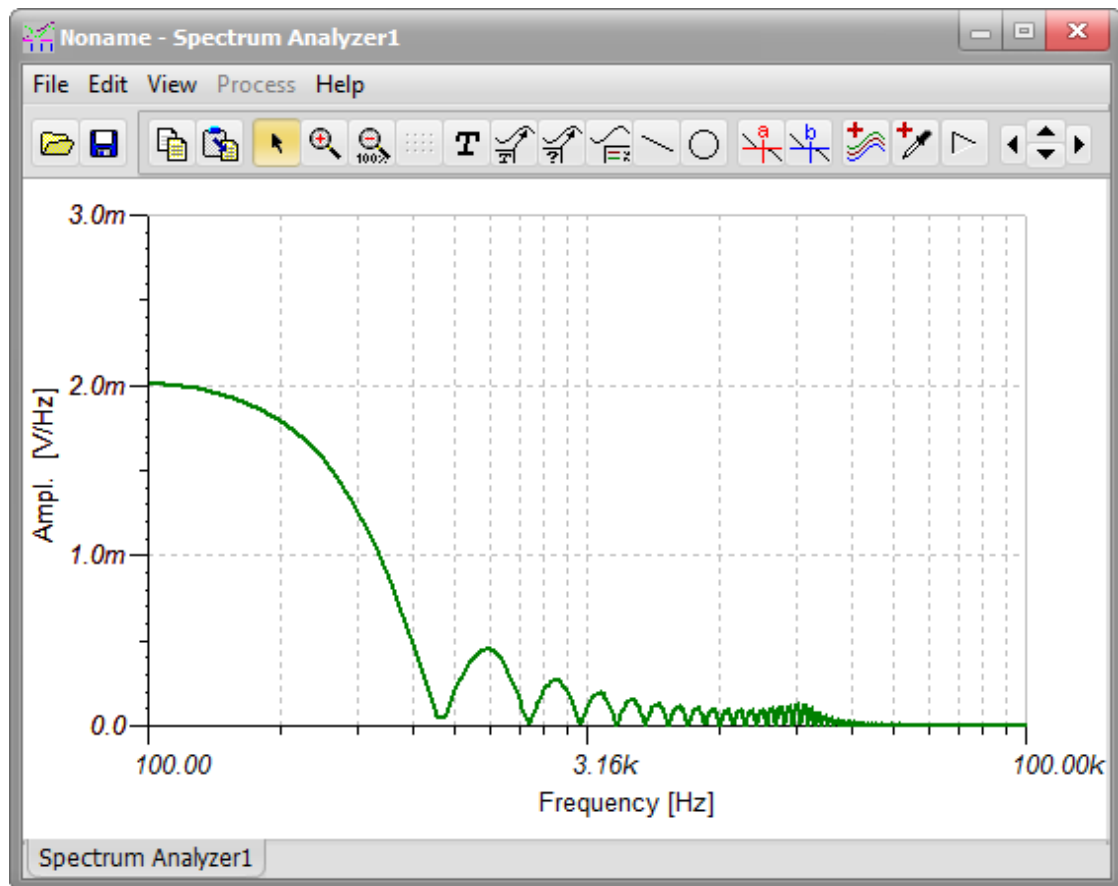




The 2m maximum is the well-known value of the  $2AT$  real amplitude density of the  $A=1V$  high and  $T=1ms$  wide pulse. You can see again the contribution of the damped sinusoidal response of the circuit to the spectrum.

Note that this function is double the amplitude of the absolute value of the complex amplitude density which you'd get using Analysis/Fourier Analysis. This is due to the fact that the spectrum analyzer shows the actual voltage you would measure. This is double because we consider both positive and negative frequencies of the complex function.

You can also export the result into a diagram - if you are using TINA - by pressing the  button on the Signal Analyzer.



If you want to see this signal in the time domain (the time response of the circuit to a pulse), invoke the real time Oscilloscope by clicking the Oscilloscope button in the TINALab window and set the following:

### Trigger

Press Mode and select Single

### Horizontal Time/Div

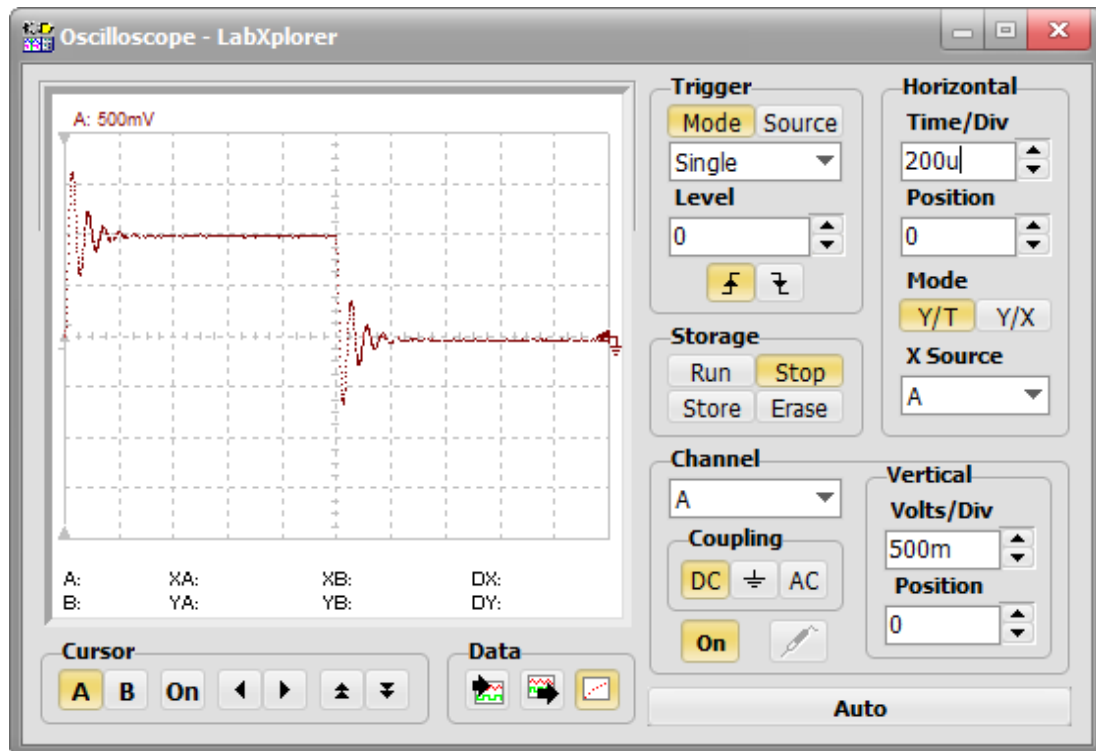
200u

### Vertical Volts/Div

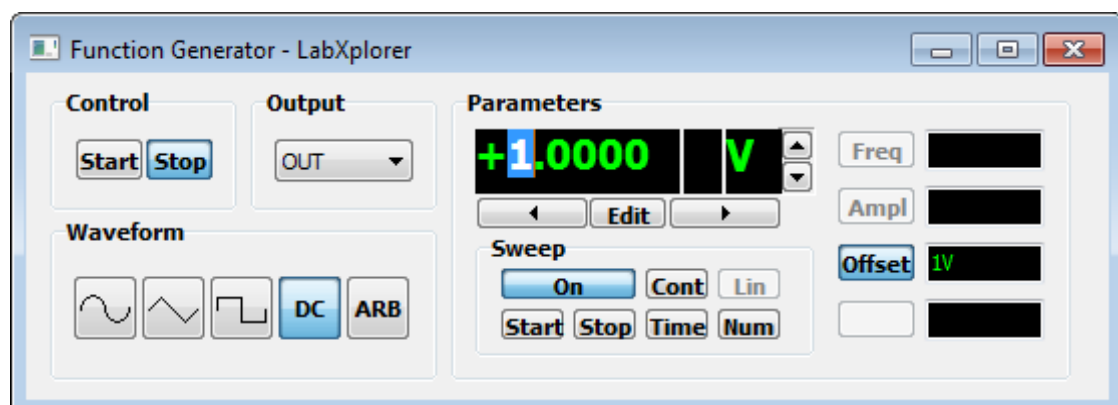
500m

Press **Run** on the Oscilloscope. It will wait for the pulse. Now press **Start** on the Function Generator. (Press **Stop** if **Start** is pressed and press **Start** again).

The time response of the circuit to a single pulse will be shown on the Oscilloscope. You can observe some noise on the signal—this is electromagnetic interference from the environment picked up by the unshielded wires we used for the measurement and the breadboard.

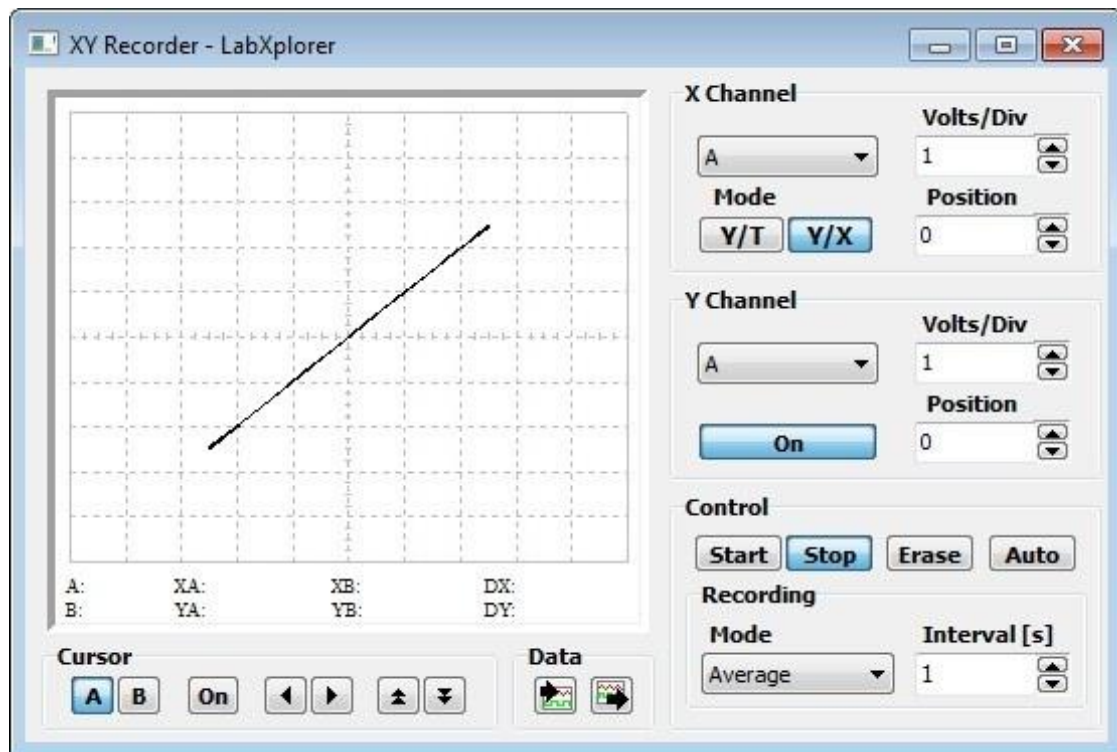


At last, click XY Recorder button to measure DC characteristic. To start the measurement first select the Function Generator. Set the waveform to 1V DC, press Sweep On and then the Control Start.

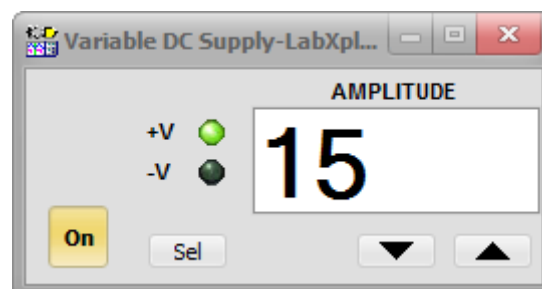


Connect Function Generator Out to Ch B, set X Channel to B, Y Channel to A, Volts/Div values to 400m and then press Run.

The measured curve will appear as you can see in the figure below.



If you want the Function Generator to control Out of LabXplorer and vary the power supply output at the same time then click the Variable DC Supply button. Use the Sel button to select if +V or -V and press On button to switch the power supplies on or off.



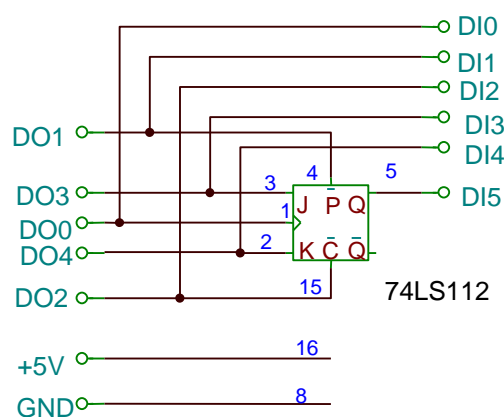
# Digital Measurements

Digital measurements use the Digital Signal Generator to excite the inputs of the logic circuits and the Logic Analyzer to capture the output logic signals. As you plan your own test cases, it is vital to consider the speed of the logic devices being tested. You should select the maximum frequency of the Digital Signal Generator to respect the switching characteristics of the logic devices under test (DUT).

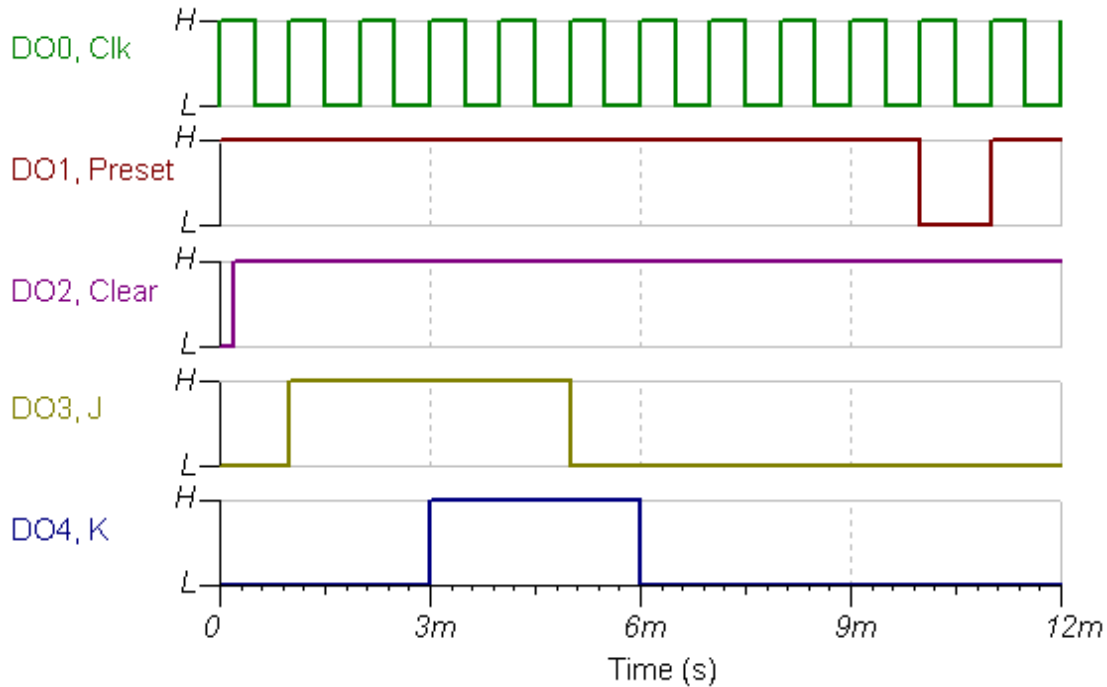
Another point is that the sampling rate of the Logic Analyzer must be a minimum of twice the frequency of the fastest signal that you are sampling. If the Logic Analyzer is not sampling fast enough relative to the DUT, information will be missed between sampling points. For example, if your system is running at 2MHz you would want to set the sampling period to at least 250ns. The sampled data is then recreated and displayed on the logic analyzer's display screen.

The faster the sampling, the better the resolution, but you should also realize that increasing the sampling rate decreases your sampling time (sample record length). Sampling time can be calculated as sample memory depth times the sampling period. For example, if you set the sampling period at 50ns, and assuming the frame memory is 64k, then it yields a 3.2768us record length.

Now, let us see in real life how to examine the operation of a simple digital circuit, a JK flip-flop. Although the example is simple, the underlying methodology is the same for more complex logic—in all cases, one must provide periodic signals (clocks) and aperiodic patterns for example inputs. On the schematic below, DO<sub>n</sub> represents the output of the Signal Generator, while DI<sub>n</sub> is the measurement point of the Logic Analyzer, where n is the channel number. (You can check the Signal Generator and Logic Analyzer without a flip-flop. In this case DI<sub>5</sub> should be unconnected.)



Let's create a set of test signals for the Digital Signal Generator and explore the behavior of the flip-flop. See the time diagram below.



Clk frequency is 1kHz and the other signal assignments are:

time, ms	Preset	Clear	J	K
0	1	0	0	0
0.2	1	1	0	0
1	1	1	1	0
3	1	1	1	1
5	1	1	0	1
6	1	1	0	0
10	0	1	0	0
11	1	1	0	0

To establish the waveform above, invoke the Digital Signal Generator. Next you need to define the two properties of the time period during which the output signals will be active. This time period is by specified by a) the minimum time step, and b) the number of time steps. In our example, the value of the Period (minimum time step) is 100us and the Length (the number of time steps) is 128. The Period property defines the duration between clock pulses, while Length defines the number of clock cycles to be generated. (At present, Digital Signal Generator operates in power of two time steps.)

Next, set the timing display left side to zero and the right side to 12.8m, the full length pattern.

In the Digital Signal Generator, the signals need to be arranged in groups. A

group is a container for any number of signals arranged in any order. It is worth arranging signals with similar functions into a group.

In our example, we will create four groups.

Group One is just for the clock (Clk) because it is an independent periodic signal. It is essential for the periodic signal(s) to be in distinct groups, because in this case we can build waveforms efficiently after one defined period using the repeat command.

Groups Two and Three are for the static control inputs (Preset, Clear). A series of logic levels individually defines these aperiodic signals individually.

Group Four is for the dynamic control inputs (J and K).

When you start the Digital Signal Generator, all the signals can be found in one group under the control called Channel. Select and Delete the initial group by pressing Del.

Now, let us create the Clk group (from the Clk signal) that will be generated on the digital output channel 0 (DO0).

Using the buttons From and To, create a new group selecting DO00 in both cases, and then enter the name (Clk) of the recently created group in the Group Label field.

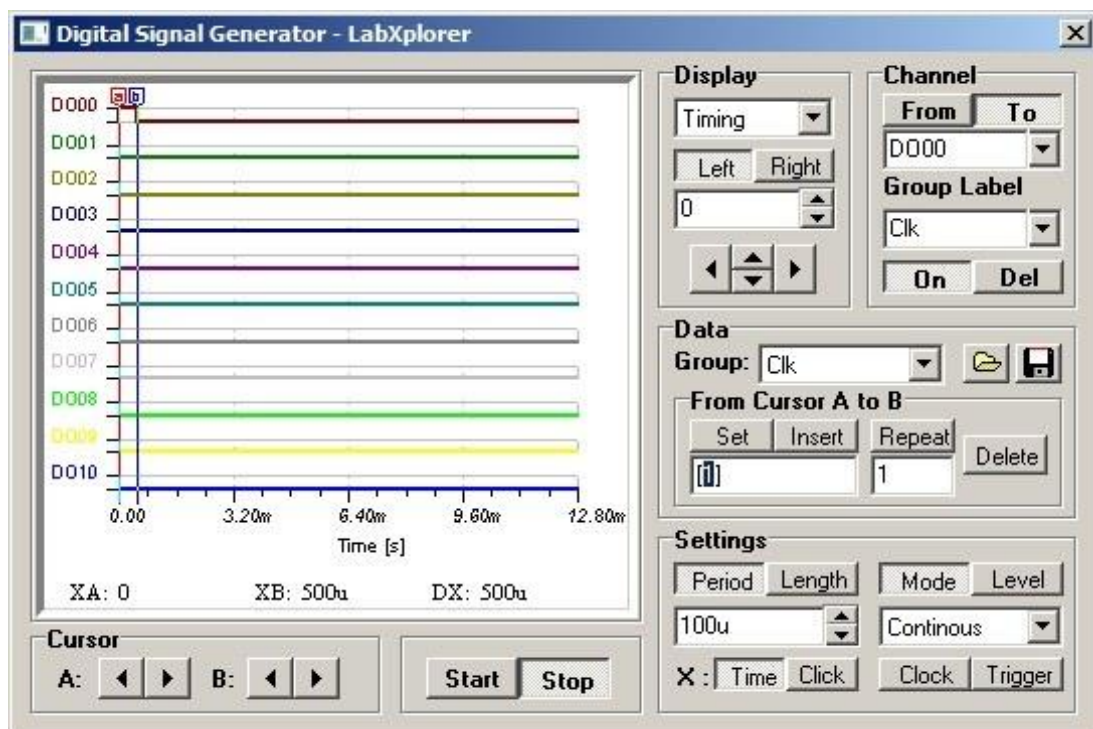
If you are successful, the new group will be seen in the Group field.

The controls of the Data field allow you to define the pattern of the signal.

The Group field automatically shows the previously created group (Clk).

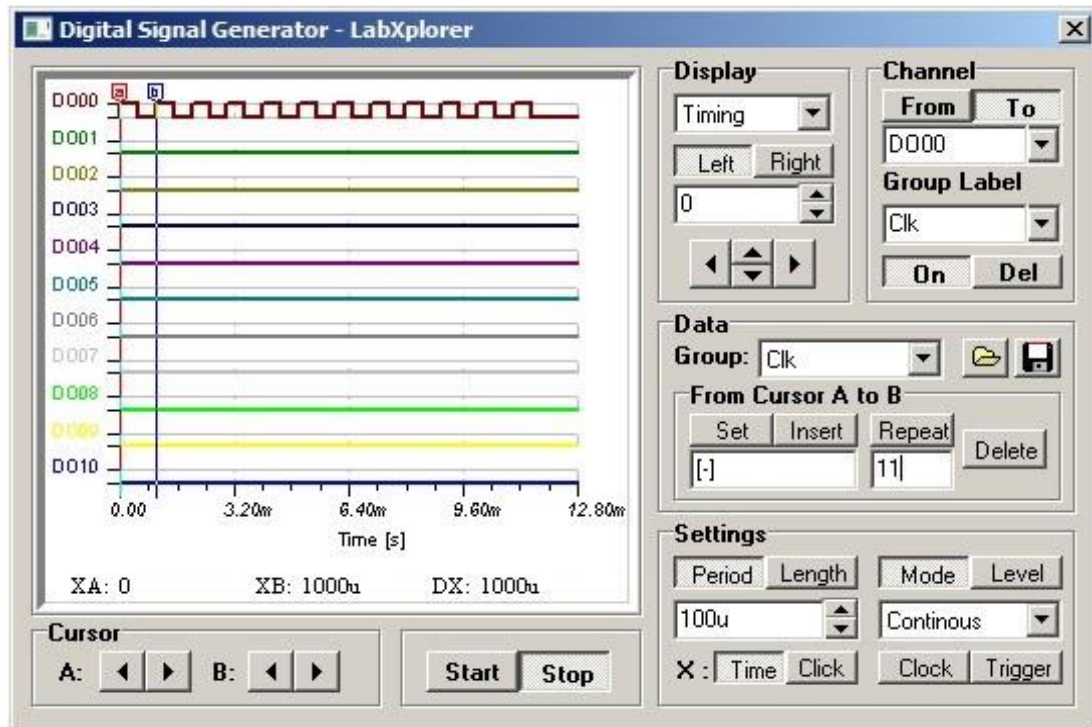
Select the Clk group in the Data field, move cursor A to 500us and cursor B to 1ms, and set the signal value to 1 in the From Cursor A to B edit field.

Finally, press the Set button to create the first period of the signal.





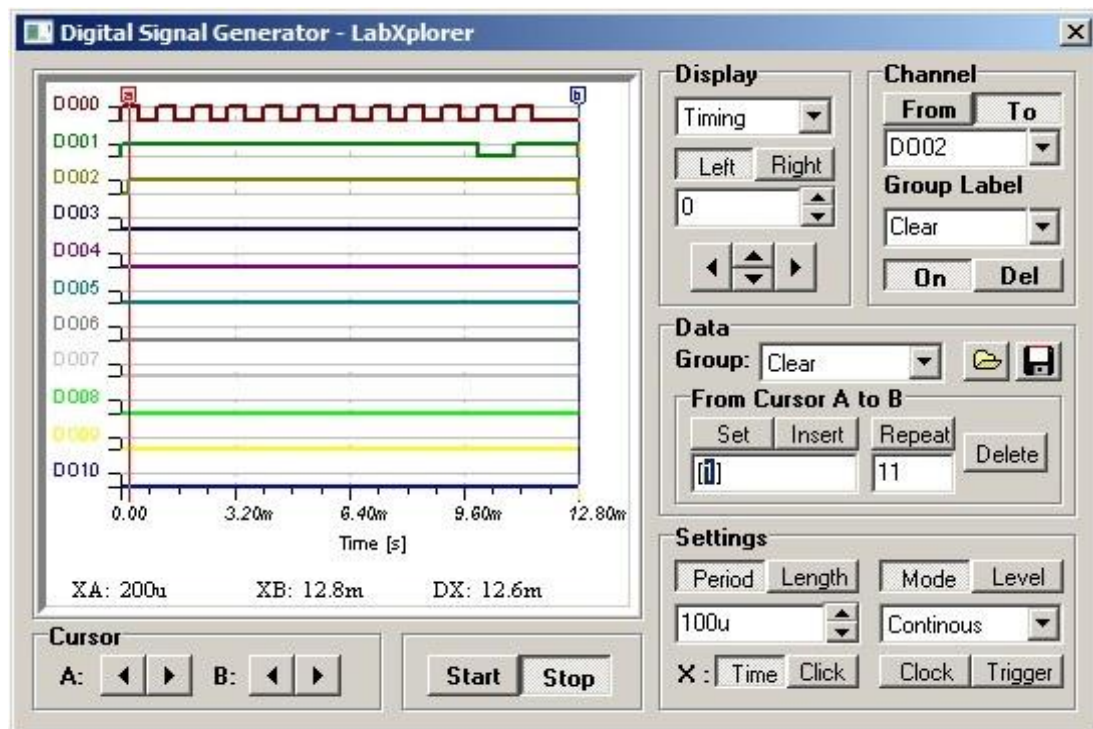
To replicate this first clock period, define the number of periods by entering 11 in the Repeat field. Before pressing Repeat, move cursor A to the beginning and cursor B to the end of the period, then press Repeat and the entire signal will be filled in.



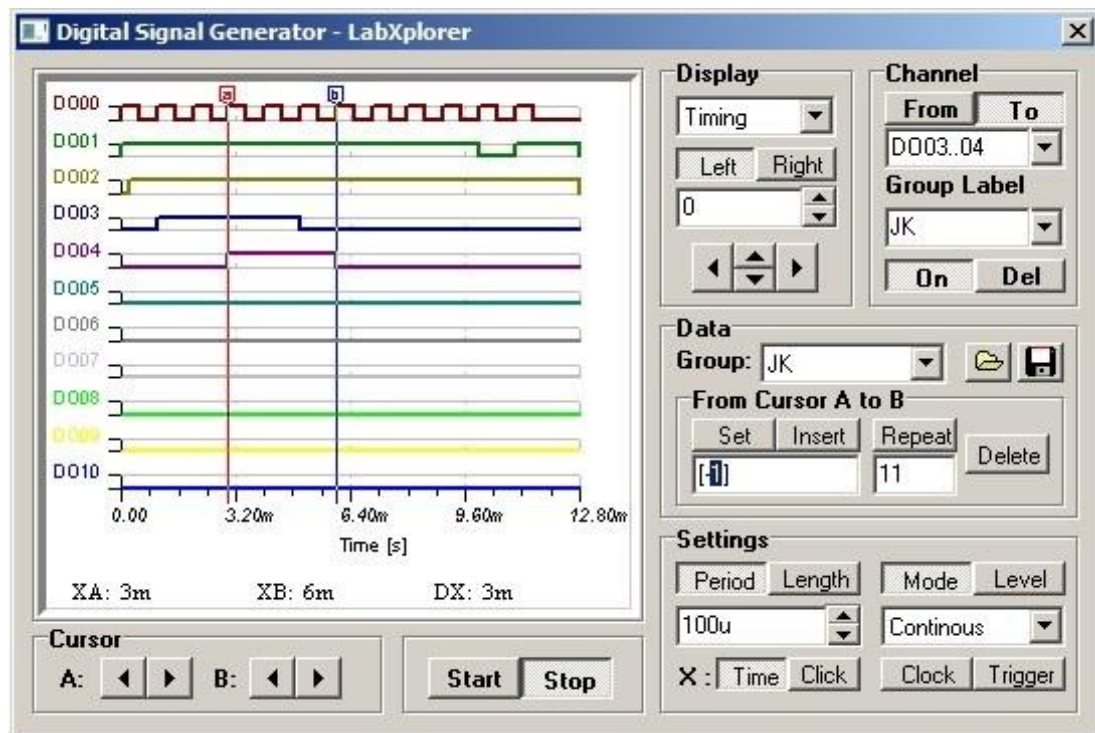
Above in a similar manner, create group Preset group (From DO01 To DO01), the Clear Group (From DO02 To DO02) and the JK Group (From DO03 To DO04). Select the Preset group in the Data field and move cursor A to 0 and cursor B to 10ms (according to the signal assignment), enter signal 1 into the From Cursor A to B edit field, and apply the Set button. Do the same with cursor A at 11m and cursor B at 12.8ms.

Create the Clear signal by selecting the Clear Group and placing the cursor A at 200us and cursor B at 12.8ms. The following window will appear, with the first three waveforms complete.





As we move to the JK Group, we encounter something new. Because the JK Group has two members, the From Cursor A to B Edit window now contains two characters as an initial value, e.g., 00. The 0 in the first position represents the initial value of signal J, while the second represents the initial value of signal K. According to the specification, if you want to set signal J to 1 within the period of 1..5ms, you have to move cursor A to 1ms and cursor B to 5ms using the cursor arrows, then set the 0 in the first position to 1, and press the **Set** button. The result appears immediately on the screen of the generator. In the same way, you can set signal K if you move the cursors to 3ms and 6ms and set the second 0 to 1.

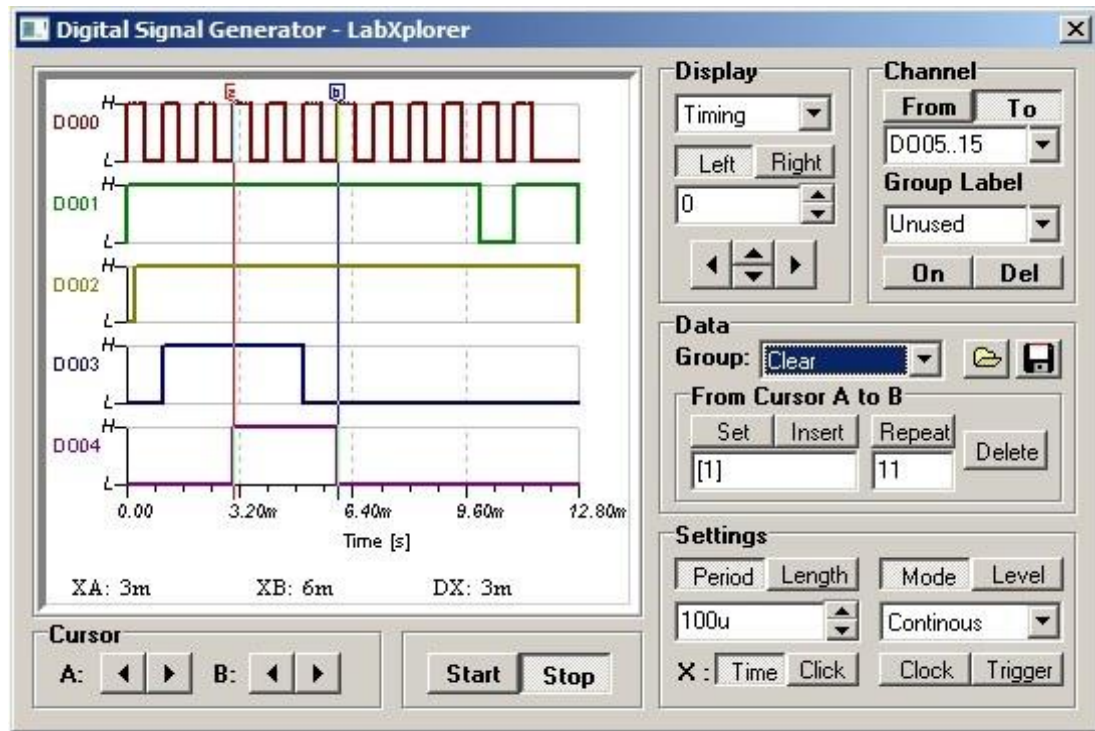


Note that after you've set the second signal (K), a '-' sign appears at the position of the first 0. This tells you that the program won't modify the state of the first signal when you modify the state of the second one. Now the state of the first signal (J), which is not constant between the actual cursor positions, will be protected. The '-' sign can be entered manually if you want to protect a signal from having its state changed when pressing Set.

If you are editing a digital signal, you will find the Insert and Delete buttons useful. You can insert a new section of arbitrary length into any point of an existing signal. If you want to insert a new section, move the cursor A to the starting point and cursor B to the end point of the section to be inserted. Then define the logic level as you have learned to do, but now instead of pressing Set, press Insert, and the new section will be inserted. Note that the Insert option affects every signal within a previously defined group. *But it does not insert steps into the signals of other groups!*

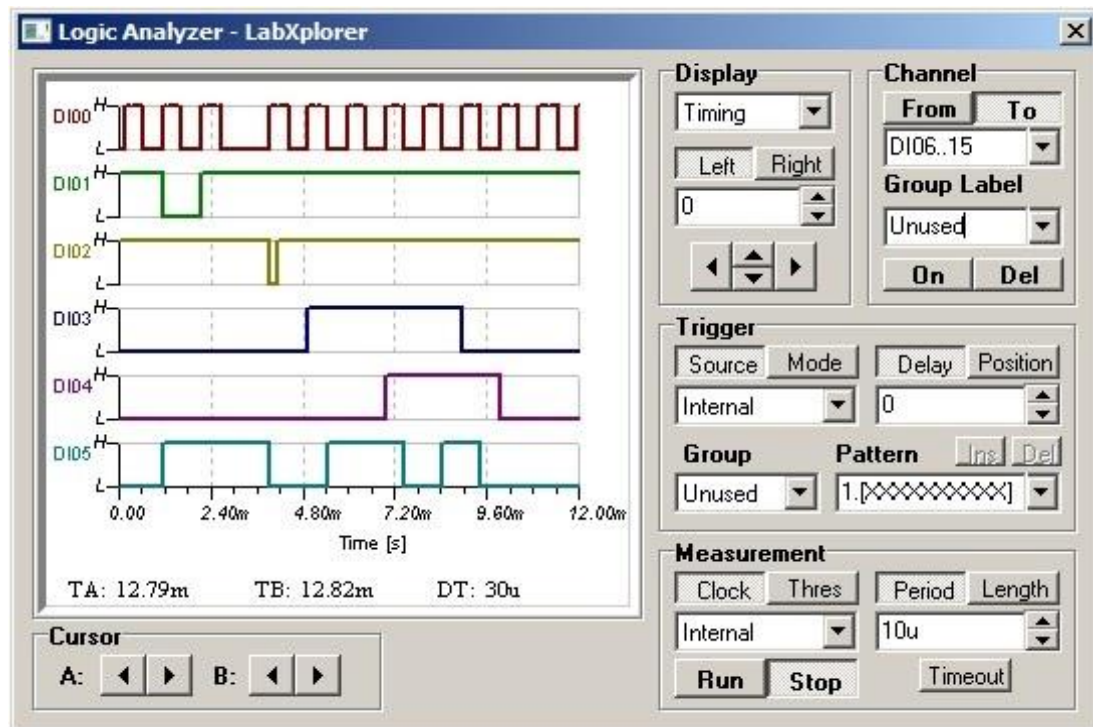
The Delete function works exactly opposite to Insert; namely, the section between the cursors will be erased and the gap filled with states pulled in from the right.

After you have finished creating the signal patterns, you may wish to remove the unused channels from the display. To do so, collect the unused channels into one group; then turn off the display of this group by pressing the On button in the Channel area. Turning on and off the display of groups is similar on the Logic Analyzer.

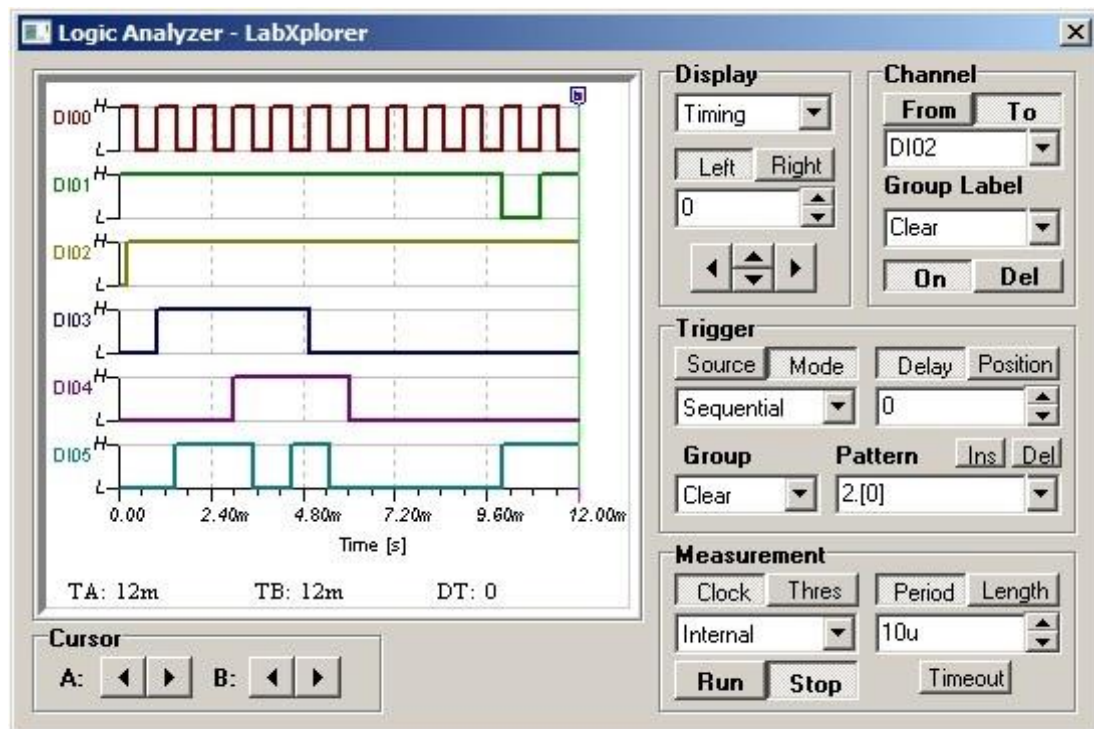


To display the digital signals, you have just created with the Digital Signal Generator, connect the digital output channels 0 to 4 of LabXplorer to the digital inputs from 0 to 4 as well as the flip-flop output to digital input channel 5 and press Start.

Click the Logic Analyzer, set the Period and Length parameters according to the previous example (10us, 1200). Start the Logic Analyzer by pressing Run, and the same signals you have set on the generator will appear on the display of the Logic Analyzer. The signals can be zoomed in (filtered) by creating groups.



To force the starting point of the drawing to the pulse of Clear, create a Clear Group (From DI02 To DI02), press Trigger mode and choose Sequential. Select the Clear Group and create two patterns (1.[1] and 2.[0]) to start acquisition from the falling edge of the signal.



# Device characteristic measurement

This chapter introduces the use of the Parameter Analyzer, the virtual instrument of the LabXplorer, which can capture the characteristic of different semiconductors.

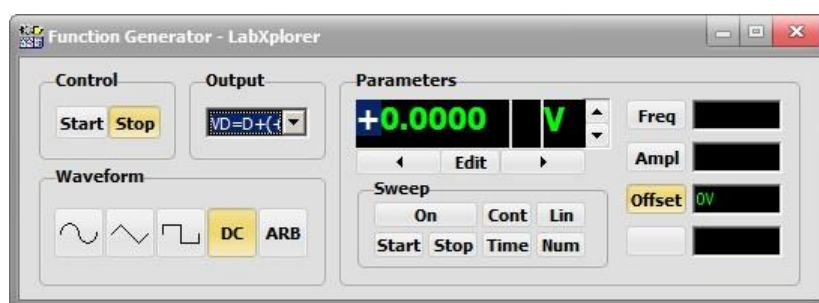
As in the previous chapters, we will explain those controls here which are necessary to understand our experience. The detailed description of all instruments is given in later chapters.

Worth noting that the device characteristic measurement is an automatic process performed by the Parameter Analyzer and Function Generator.

First, the component under measurement must be connected to the D–, D+ and/or B (base or control) sockets of the LabXplorer depending on the number of the terminals of component. If we have a two-terminal component like diode, D– and D+ are used. The third terminal of the transistors can be plugged in socket B.

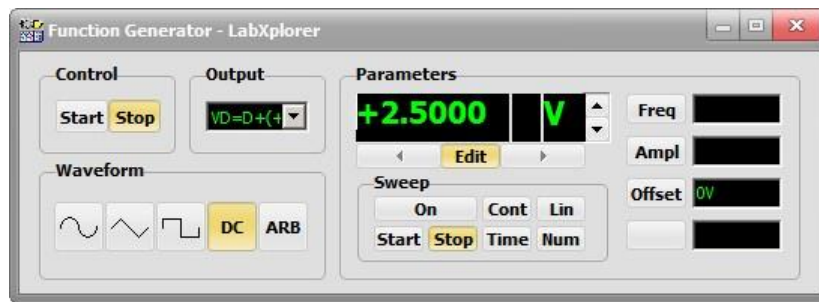
Now, let's connect an LED, - the anode to D+ and cathode to D–, using cables provided with the instrument- then start Parameter Analyzer and select 'Diode ID(VD)' measurement mode.

Now, let's click on the Source button of the Analyzer to set the sweep voltage on the diode. The Function Generator will appear.

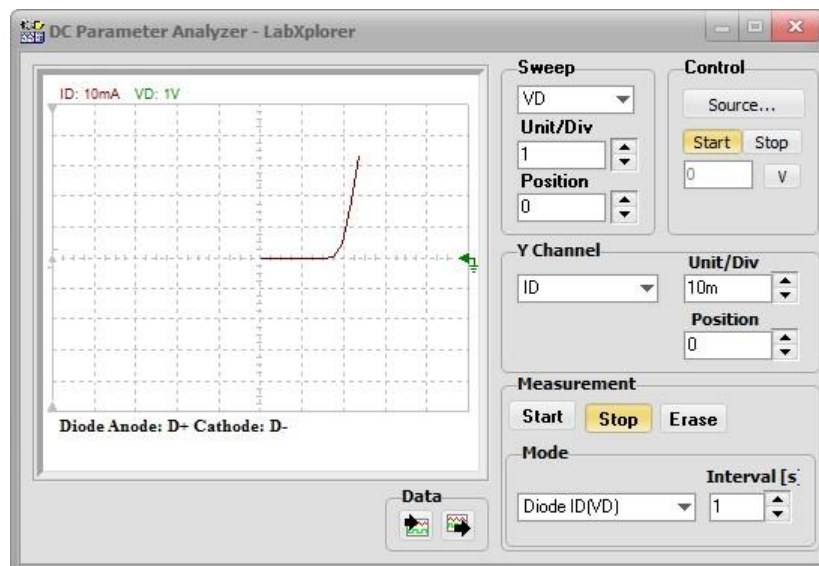


Check if the output channel has been already set to  $VD=D+(+V)$  with DC waveform as above, let the sweep start value 0V and then click the Stop and the Edit buttons. Enter the value +2.5.





Now, click the Start button of the Analyzer. The Analyzer will change the generator output during the sweep from the Start to the Stop value and measure the LED current back. The sweep channel Unit/Div value will be automatically adjusted to the sweep range. The measured curve will appear as you can see in the figure below.



Now, click the Data Export button to move the curve into the Diagram Window.

# Test Card Experience

DesignSoft manufactures ready-to-use test cards covering the whole range of analog and digital and programmable electronics. The test cards are plugged into the slot on the front of LabXplorer and allow you to simulate, measure, and troubleshoot. (See <http://www.tina.com> for the latest list of test cards.)

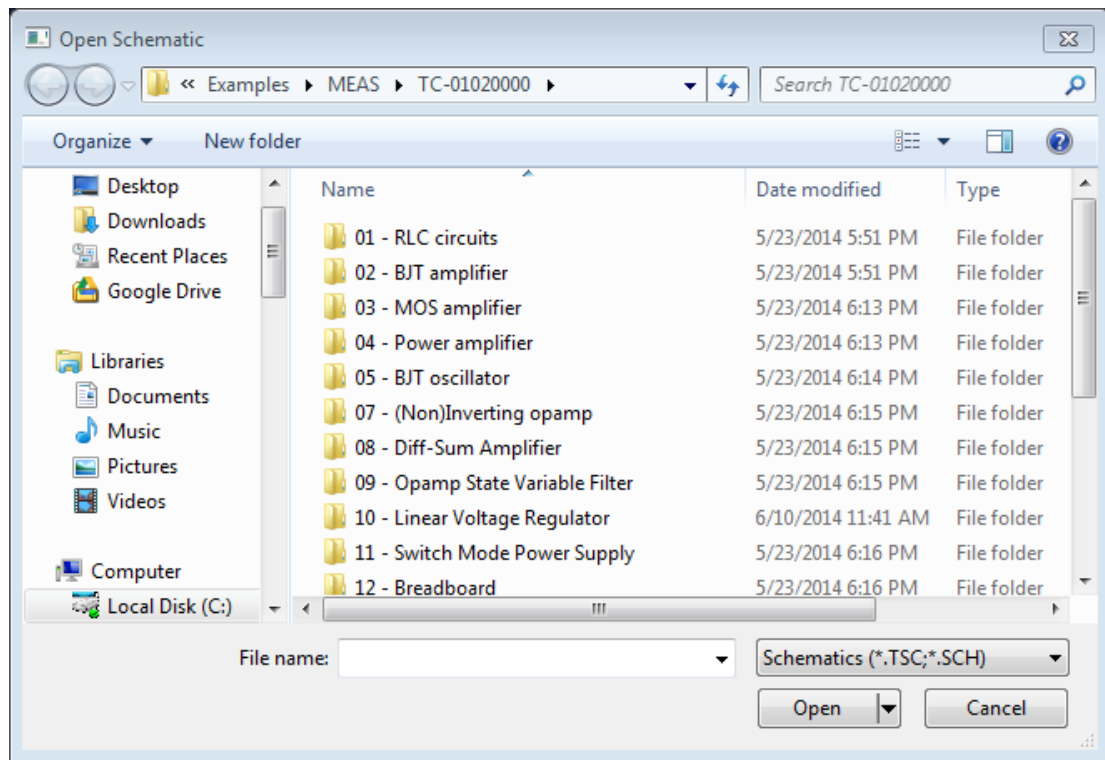
The hardware and software components can be configured in several ways. Using LabXplorer with TINA or TINACloud, DesignSoft's popular circuit simulation programs, gives you the unique capability to have circuit simulation and real-time measurements in the same integrated environment. This is invaluable for troubleshooting; furthermore, it brings your designs to life by comparing simulated and measured results.

A Windows workstation with any TINA software considered as a local environment. The instrument is connected directly to the computer, the software controls the measuring and displaying the process.

In remote measurement, the LabXplorer is connected to a Windows Server typically or a workstation. The clients can access the device running TINACloud in a browser. At the same time TINACloud schedules requests of the clients for the LabXplorer resources (sharing).

Now try your hand at measuring test circuits with TINALab Basic.

Now, firstly plug a board into the Test Board slot - the Analog Experimenter Board is convenient for a quick test - then start the program, click Testcard from the menu, and choose Open Testcard... The program will automatically recognize the board and opens the folder containing the circuit schematics on board.



Double click the folder where groups of circuit schematics are and do on the specific file to measure. Now, open TC-01020103 RLC circuits from the '01 – RLC circuits' folder.

The program reads the selected schematic file, the simulation parameters and the help of the file name code configures the hardware.

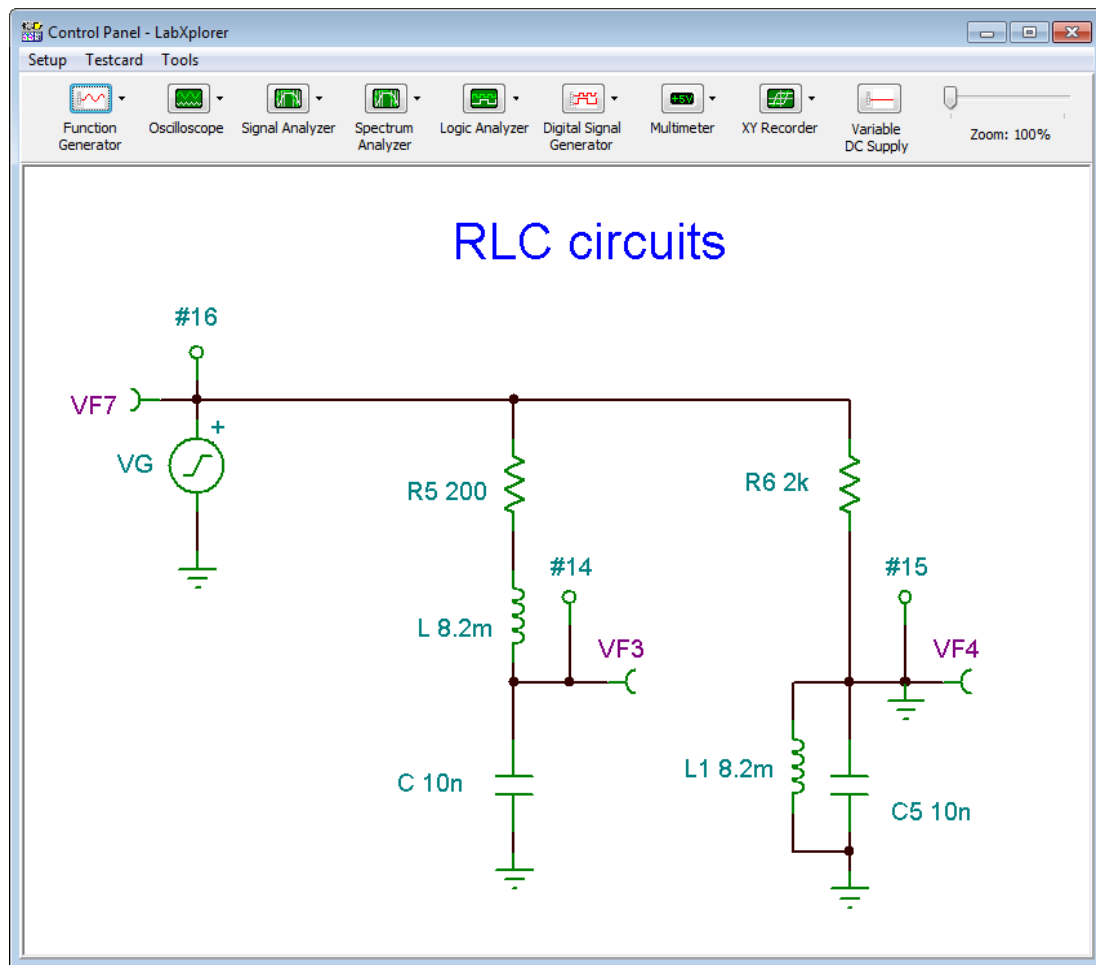
The file name code is typically TC-01xxyyzz..., where

xx the identifier type of the circuit board (02 – analog circuits with breadboard)

yy the code of the circuit group

zz the configuration code within the same circuit group



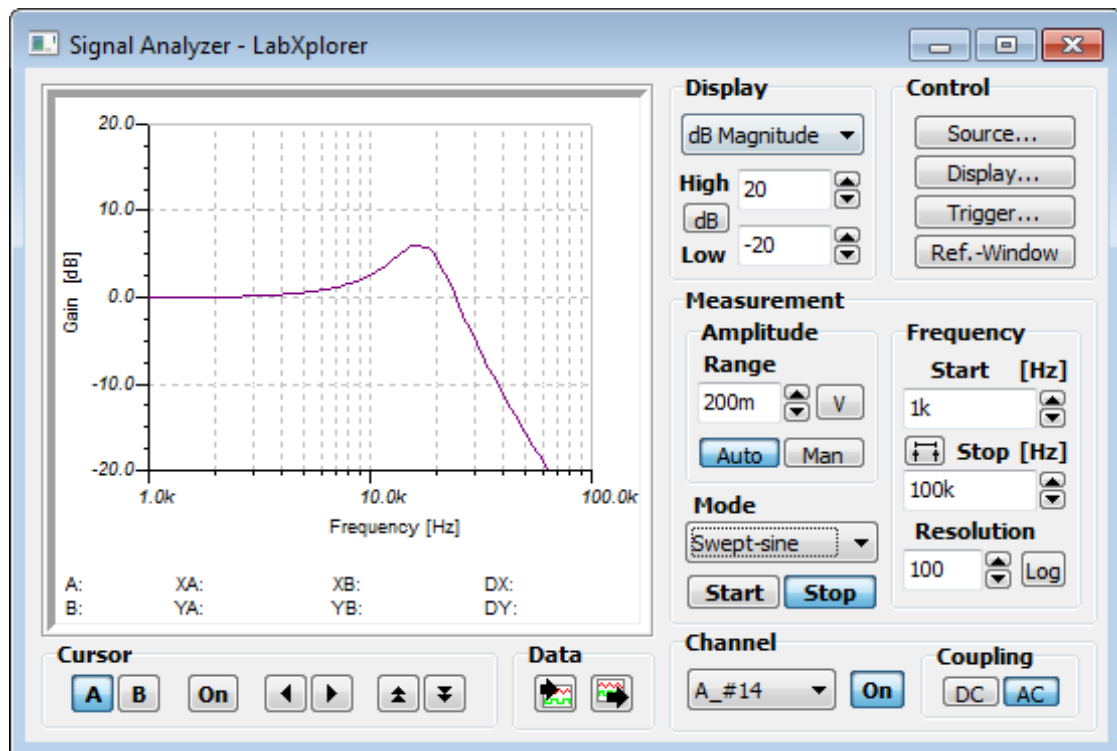



The nodes marked # are real time measurement points (numbered from 1 to 16) and are hard wired to the selected inputs of the slot. (e.g., #3 means input channel 3). The input channels can be analog or digital depending on the circuit details.

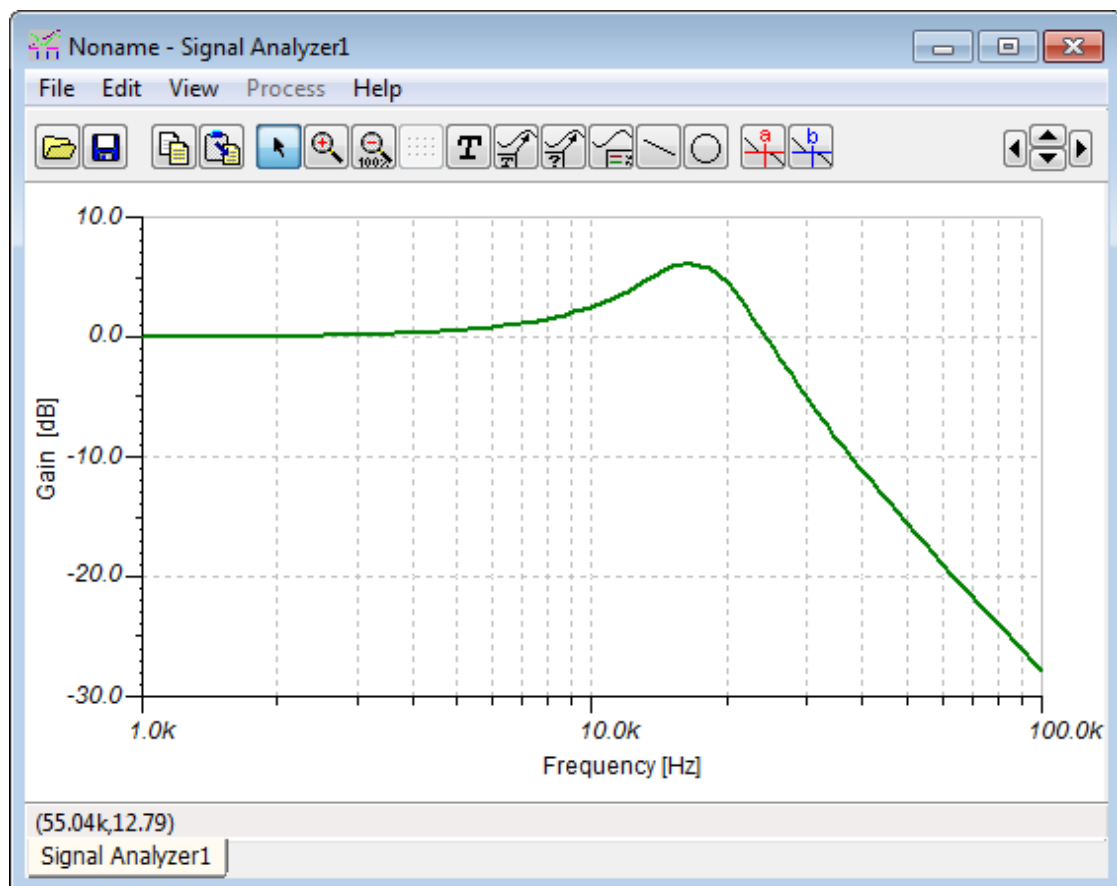
From the viewpoint of LabXplorer, the output in an analog case is the voltage generator on the schematic, while the outputs in digital cases correspond to clock and pulse sources.

All source parameters are copied from the schematic settings and transferred into LabXplorer at the start of the measurement. (For more information on parameter transfer, see chapter Generator matching.)

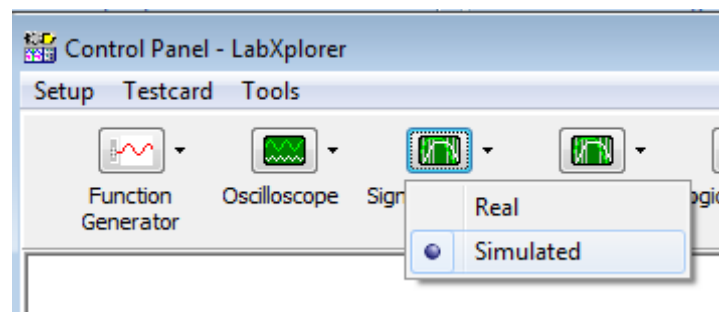
Next, choose the Signal Analyzer from the instruments and measure the transfer characteristic. Switch channel A off by pressing the On button next to the channel selection edit box, choose A\_#14 (or B\_#14)–the output of the serial RLC network, – check display and frequency settings then press Start. A short calibration cycle follows, after which the measured curve will soon appear.



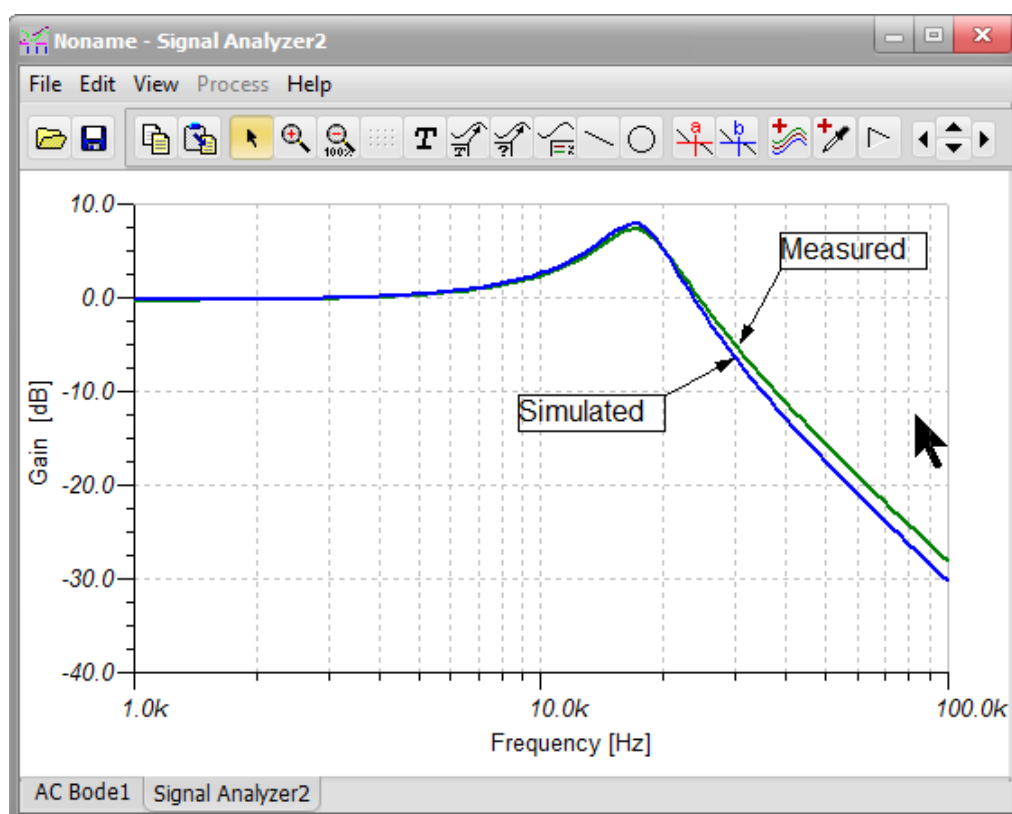
We can export the resulting curve into the Diagram Window by pressing the  button.



To compare the measured data to simulation results, choose the Simulated Signal Analyzer.



Leave the settings as they are, press the start button, and then double click on the resulting curve to set its properties (color, etc.) to be different from the measurement curve, and copy it onto the clipboard. Switch back to the measurement diagram and paste the clipboard content into it. Now the measured and simulated curves are both in one diagram, making comparison easy. The slight dissimilarity between measured and simulated curves is mainly due to component tolerances.



Note: To save the diagrams, click File.Save as... or File.Export for different file formats.

# Generator matching

Generator matching is an automatic 2-way process synchronizing the analysis' parameter and that of signal sources on the schematic with the real setting of the hardware.

The Generator matching can be switched on or off by checking the appropriate field under Options... of the T&M menu point.

Generator matching is a file property. This setting is always saved with the schematic and read back when you open a saved file.

When you open a file and the Generator matching option is on (see T&M.Options...) then the output(s) of the LabXplorer will be set according to the file analysis settings. For example, one of the Voltage Generator signal settings as well as the AC transfer analysis parameters will be transferred to the Function Generator of LabXplorer. After that, if we change the settings of the Real-time Function Generator then that of the Voltage Generator and analysis parameters will be adjusted automatically.

The capabilities of the real hardware let the Voltage Source, Voltage Generator, Current Source, Pulse Source, Clock schematic symbols (altogether signal sources) play role in Generator Matching.

For the analog signal sources, the following rules are applied in the synchronizing process:

First, if any symbol label matches one of the names of the Function Generator outputs and the amplitude unit is also identical then the specific signal source will be synchronized. If symbols with the same labels exist then only one of them (not specified which) will be synchronized.

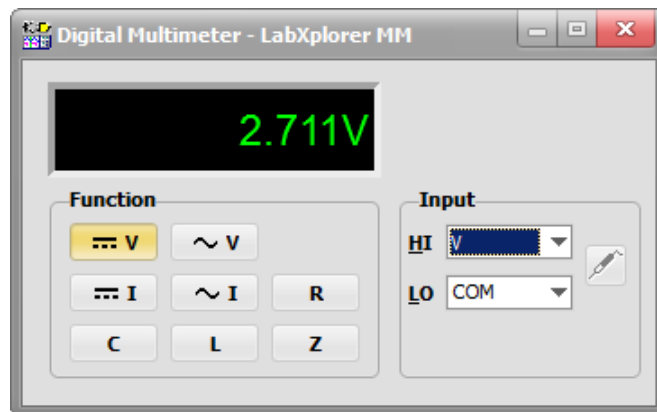
Secondly, if the symbol label does not match any of the names of the Function Generator outputs, but the amplitude unit is identical then one generator (not specified which) will be synchronized to the output of the Function Generator (Out).

While the main parameters of the matched Function Generator channels (e.g.: amplitude, offset, ...) are extracted from the signal sources, their sweep values will be determined by either that of the AC or DC transfer characteristic found under the Analysis menu point.

The successfully matched output(s) of the generator will be started at the beginning of the measuring phase. If we open and start a Real-time virtual Oscilloscope or other capturing instrument and matching is active then the matched output of the Function Generator will be started. When multiple outputs must be operational then they will be started up in pre-defined order; basically, first the DC outputs and finally the Out.

# Virtual Instruments in LabXplorer

## Multimeter



Use the digital multimeter to measure DC and AC voltage, DC and AC current, resistance, capacitance, inductance, and impedance.

If your LabXplorer does not have a Multimeter Extension Card, you can only measure DC and AC voltage. In this case, the Ch A and Ch B inputs can be used as input HI, while the LO input is always tied to the analog ground. If you have card inside, the connections are as follows:

	HI	LO
V, R	V/ $\Omega$	COM
I	I	COM
C, L, Z	D+	D–

### Measurement function controls

The function control buttons allow you to select measurement functions.

	DC Volts
	AC Volts
	DC Current
	AC Current
	Resistance
	Capacitance
	Inductance
	Impedance

## Range controls

The multimeter automatically selects the range using autoranging, or you can select a fixed range using manual ranging when you measure on either Ch A or B connectors.

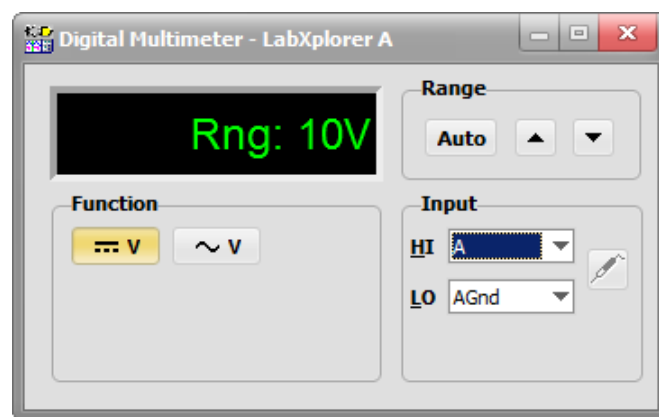


Toggles between autoranging and manual ranging.

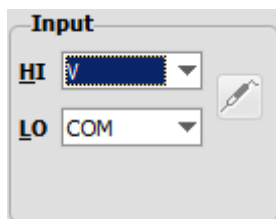
Selects a higher range and disables autoranging

Selects a lower range and disables autoranging.

If you change the range manually, the instrument displays the actual range for a moment.



## Input select



The inputs are used to connect the Multimeter to the voltage, current, resistance, capacitance, inductance, and impedance to be measured. The HI input is positive with respect to the LO terminal.

# Function Generator

The function generator is a multi-purpose generator which can be used as:

- **a reference source**

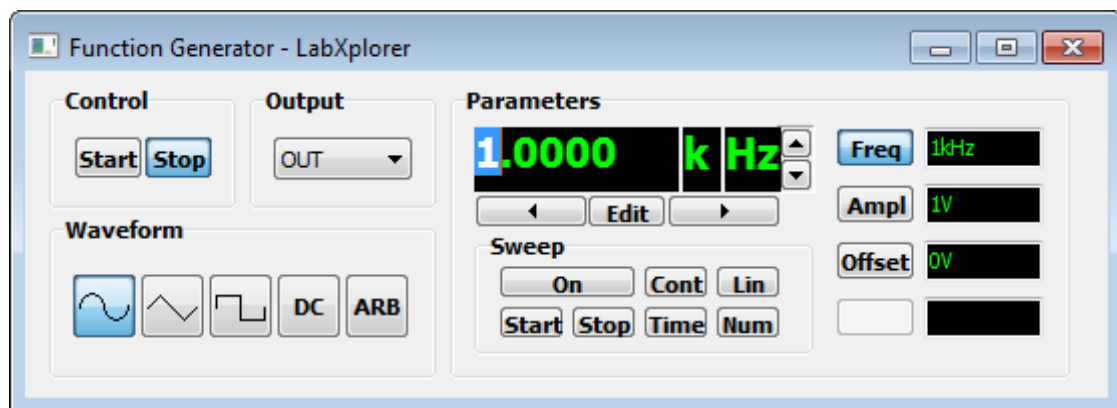
Produces a sine wave at a specified frequency, amplitude, DC offset, and phase.

- **a function generator**

Produces various waveforms at a specified frequency, amplitude, DC offset, and phase.

- **a sweep generator**

Produces logarithmic and linear frequency sweeps.



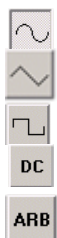
## Control



Allows you to start and stop the generator.

## Waveform

Selects a waveform



Sinusoidal

Triangle

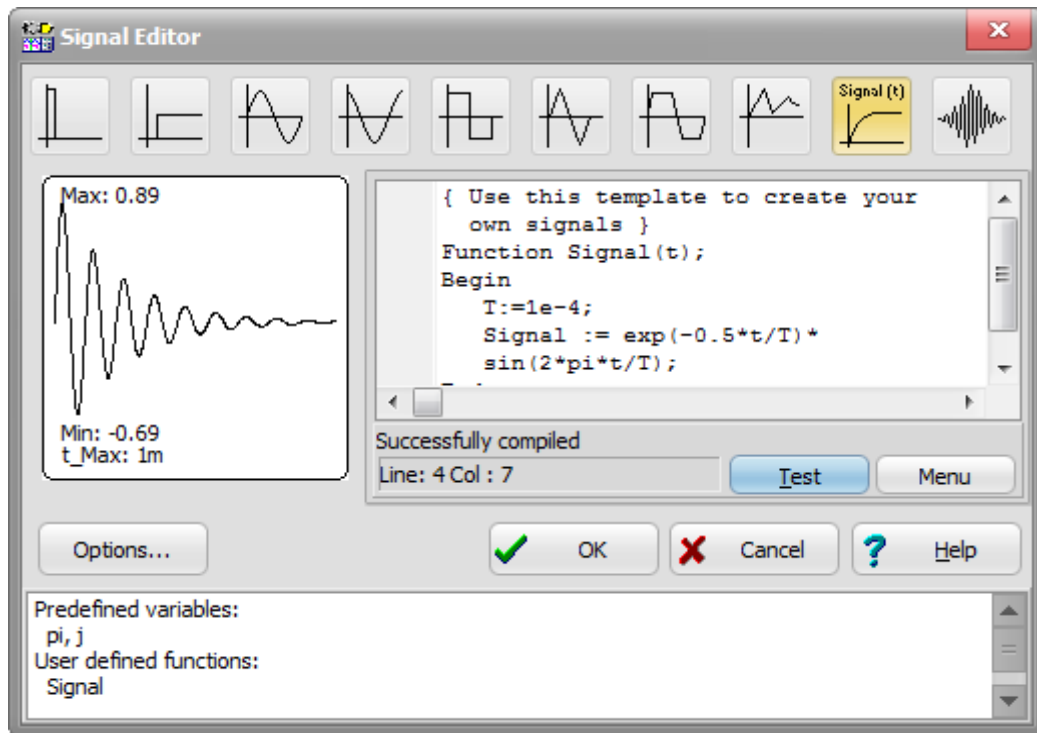
Square

DC Level

Arbitrary

Check the Specification chapter for maximum frequencies. The program constrains the entered values to the range of the LabXplorer Specification.

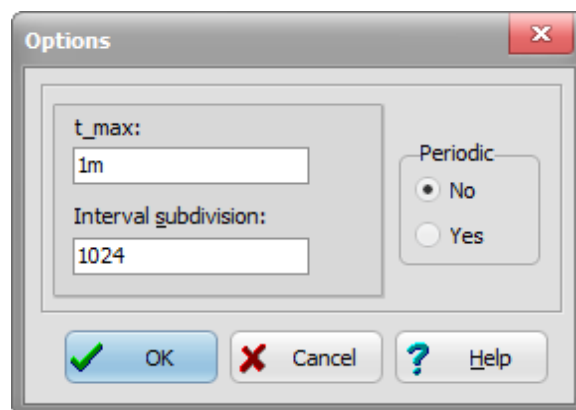
All the standard waveforms and their DC Level are set automatically according to the parameters you choose. When you choose Arbitrary waveform, the following window will appear:



The arbitrary signal must be defined as a mathematical expression. The expression has to be entered as a Function in the Signal Editor's built-in interpreter.

The Function has one input parameter:  $t$ , which denotes time. The function returns the value of the signal at the  $t$  parameter value. The interpreter evaluates the Function in the interval from 0 to  $t_{\text{max}}$  in steps defined by the **Interval subdivision** parameter described below.

To set  $t_{\text{max}}$  or change the Internal subdivision parameter, press **Options...** and you will see the following window:





Here you can choose between Single Shot and Periodic modes. Select Yes in the Periodic box.




If you are using the Function Generator in Periodic mode, the waveform defined between 0 and  $t_{\text{max}}$  will be repeated with the frequency given by the Freq parameter (described below). In Periodic mode, the value of the Interval subdivision must be dividable by a power of two; e.g., 256, 512, 1024 etc.

If you wish to change the frequency of the arbitrary signal, you don't have to invoke the Options panel—just change the Function Generator's Freq parameter.




## Parameters



Selects the parameter.

	Freq
	AC amplitude
	DC amplitude or AC offset



Edits the value of parameters.

 Edit mode - if pressed, you can enter digits from the keyboard. If not pressed, then use the following controls.

	select the previous/next digit
	step to a higher/lower value of the selected digit

To change the prefix, click into the edit box before the measuring unit name and use buttons as above.

The program constrains the entered values to the range of the LabXplorer specification. (See specification.)

## Sweep



This group gives you control over the frequency versus time operating mode if the signal is AC. If the signal is DC then the DC amplitude (offset) can be controlled. With sweep off, frequency/offset is constant—the sweep function is disabled. The buttons perform the following functions when sweep is active.

<b>On</b>	switch on sweep
<b>Cont</b>	continuous or single sweep mode
<b>Lin</b>	linear or logarithmic sweep
<b>Start</b>	start frequency/offset
<b>Stop</b>	stop frequency/offset
<b>Time</b>	sweep time
<b>Num</b>	number of steps

## Outputs

Select the output controlled. The default is the Out connector of the LabXplorer.

+V and -V is the variable DC supply.

B[V] and B[A] are mutually exclusive outputs labelled B on the front. The DC Parameter Analyzer mainly use them.

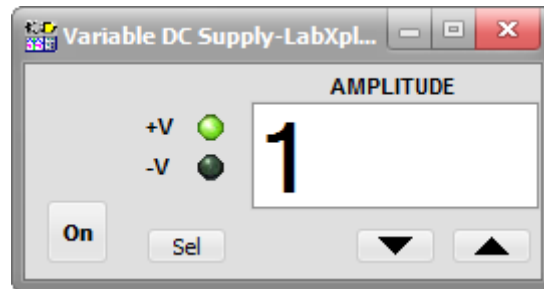
The Parameter Analyzer can change the output configuration of the Function Generator. Depending on the selected measurement mode of the running Analyzer the output channel name will be renamed. See more at the description of the DC Parameter Analyzer.

If your LabXplorer has a Multimeter Extension Card then it will control the Out channel automatically when it is measuring capacitance, inductance, or impedance value of a component connected to D+ and D-. In this case a sinusoidal signal is transferred to the D+ connector internally.

## Variable DC Supply

Change the output voltage of the DC supplies. +V allows output voltage from 0V to +15V. -V from -15V to 0V.

The outputs have the same name on the front panel (+V, -V) or they might be labelled +15V and -15V.



Sel – select supply output

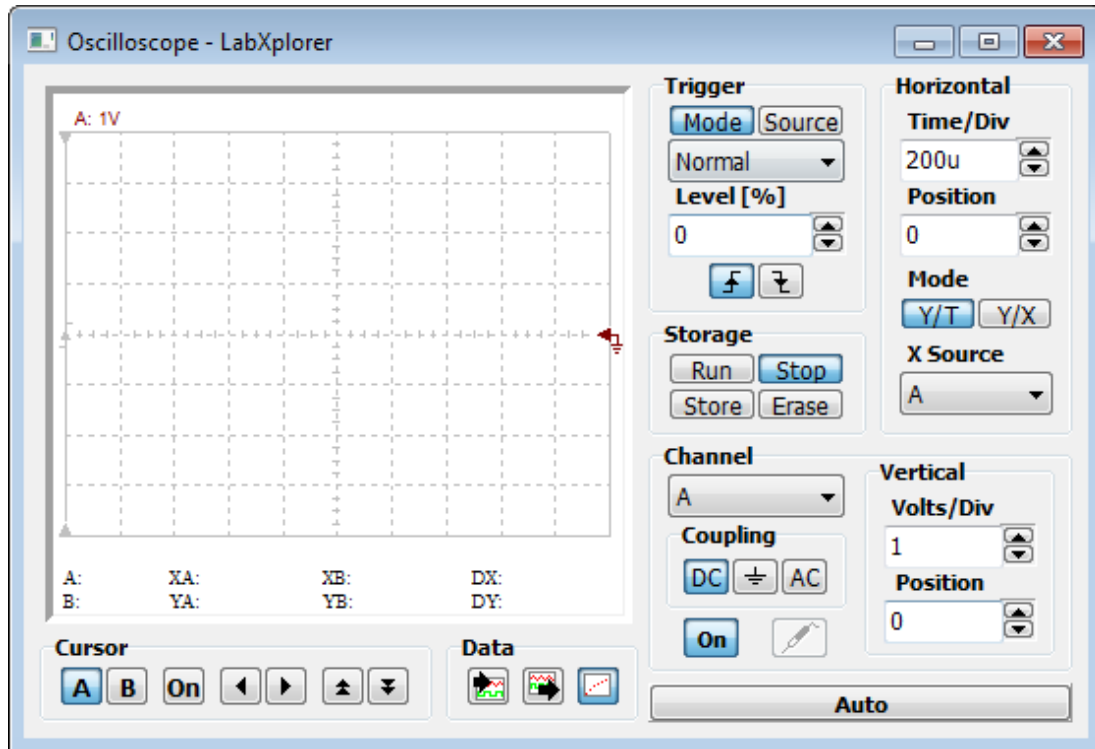
On – switch the selected output on or off



- step to a higher/lower value

## Oscilloscope

The oscilloscope displays an electrical waveform as a function of time on its screen. TINA's virtual oscilloscope has more input channels than a conventional oscilloscope, so more signals can be displayed simultaneously.

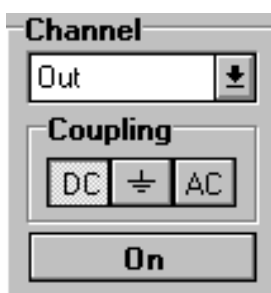


## Connectors

CH A and CH B: Input connectors are for waveform display.

EXT TRIG: Input connector is for an external trigger source. Use Trigger Source External.

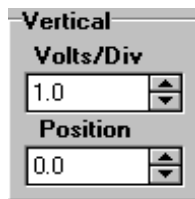
## Vertical Control



Selects the input channel.

Changes the input coupling for the channels. You can use the Trigger Coupling option to determine which part of the signal will pass to the trigger circuit. This can help you attain a stable display of the waveform.

Turns the channel on or off.

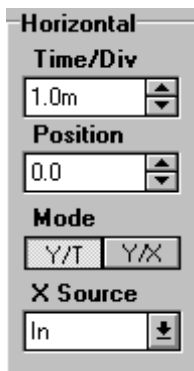


Changes the vertical scaling in a 1-2-5 step sequence.

Positions the waveform vertically.

Coupling, vertical sensitivity, vertical position, channel on/off, and coupling are entered and stored for each input source.

## Horizontal Control



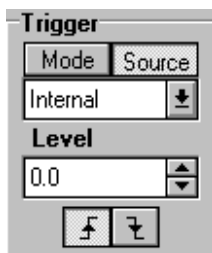
Changes the time base in a 1-2-5 step sequence.

Adjusts the horizontal position.

Changes the display to a volts versus time or a volts versus volts display.

Selects the X axis source channel in X-Y mode

## Trigger Control



Allows you to select a Trigger Mode to define how the oscilloscope acquires data when it does not detect a trigger condition. The modes are Single, Normal, Auto, and Free run.

### Single:

When the oscilloscope detects a valid trigger, it acquires a single waveform then stops. Use this mode to record transient (non-repetitive) waveforms; such as to capture a relay back electromagnetic pulse or the voltage of a discharging capacitor.

### Normal:

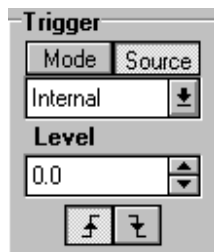
Updates displayed waveforms only when the oscilloscope detects a valid trigger condition; the oscilloscope displays older waveforms until the oscilloscope replaces them with new ones. Use this mode when you want to see only valid triggered waveforms; when you use this mode, the oscilloscope does not display a waveform until after the first trigger.

### **Auto:**

The oscilloscope automatically forces triggering periodically if it does not detect a trigger within a certain amount of time based on the TIME/DIV setting. You can use this mode in many situations, such as to monitor the level of a power supply output. Use this mode to let the acquisition free run in the absence of a valid trigger.

### **Free run:**

The oscilloscope automatically acquires waveforms, ignoring trigger events.



Source option selects the signal that the oscilloscope uses as a trigger. The source can be Internal, Ch A, Ch B, External, Generator.

Ch A, Ch B, External are the signal source connected to the BNC connectors

Generator: internal loopback to the Function Generator Output

When you use an Edge trigger, the Level field sets the amplitude level the signal must cross to cause an acquisition.

Selects either the rising or falling edge of the signal to trigger the oscilloscope.

## **Storage Control**

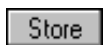
There are four storage keys to change the operating mode of the oscilloscope.



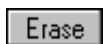
The oscilloscope acquires data and displays the most recent trace.



Freezes the display.



The oscilloscope acquires data and displays the most recent and previously acquired waveforms together.



Clears the display.

## Autoscale Control




The oscilloscope has an Autoscale feature that automatically sets up the oscilloscope to best display the input signal.


## Display Area

In addition to displaying waveforms, the display provides cursor details. You can use two cursors to measure amplitude and time values as well as their delta.

 select cursor

 on/off cursor

 step cursor

 step another curve

## Signal/Spectrum Analyzer

Traditionally, when you wanted to look at an electrical signal, you used an oscilloscope to see how the signal varies with time. This is very important information; however, it doesn't give you the full picture. To fully understand the performance of your device/system, you will also want to analyze signals in the frequency domain. This is a graphical representation of the signal's amplitude as a function of frequency. The spectrum analyzer is to the frequency domain as the oscilloscope is to the time domain.

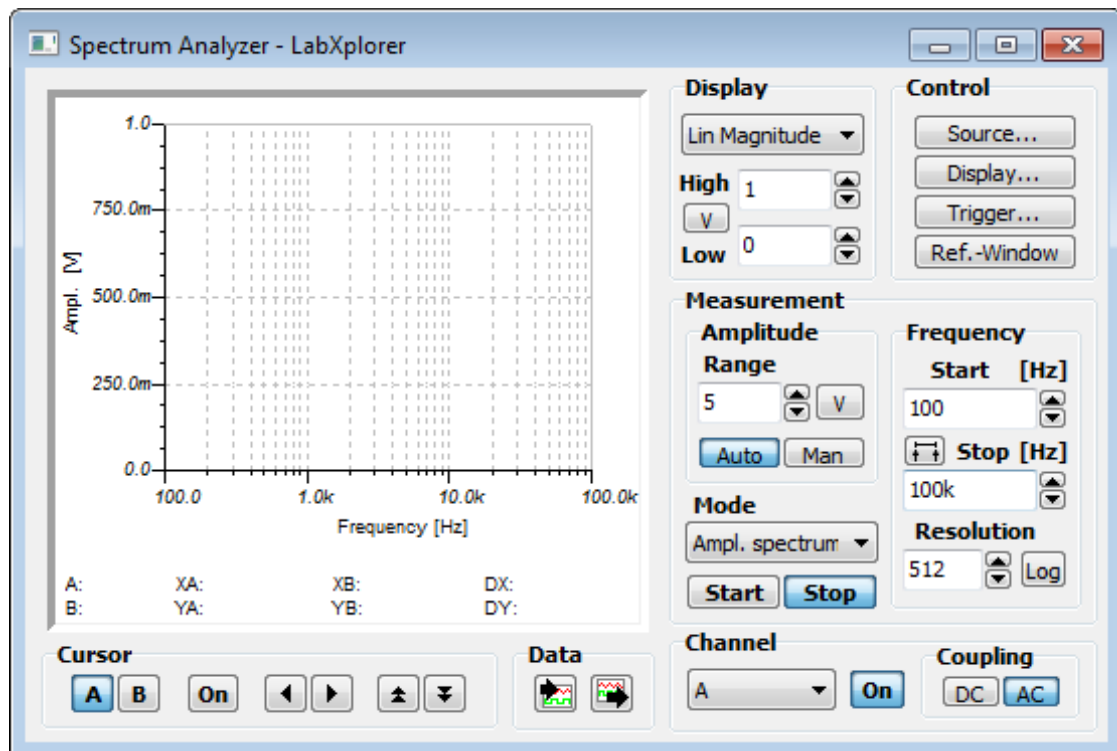
There are basically two ways to make frequency domain measurements (what we call spectrum analysis): Fourier transform and swept-tuned.

LabXplorer's Spectrum Analyzer uses the Fast Fourier Transformation (FFT), while the Signal Analyzer is a swept-tuned device. They can often be used interchangeably.

The most common type of spectrum analyzer is the swept-tuned receiver; it is also the most widely accepted, general-purpose tool for frequency-domain measurements. Basically, these analyzers “sweep” across the frequency range of interest, displaying all the frequency components.

The Fourier analyzer (Dynamic Signal Analyzer) takes a time-domain signal, digitizes it using digital sampling, performs the mathematics required to convert it to the frequency domain, and then displays the resulting spectrum. It is as if the analyzer is looking at the entire frequency range at the same time using parallel filters measuring simultaneously. It is actually capturing the time domain information which contains all the frequency information in it. With its real-time signal analysis capability, the Fourier analyzer is able to capture periodic as well as random and transient events. It also can provide a significant speed improvement over the more traditional swept analyzer and can measure phase as well as magnitude. However, it does have its limitations, particularly in the areas of frequency range, sensitivity, and dynamic range.



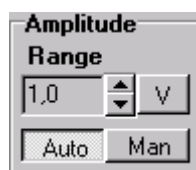


To set up the input channel



These controls allow you to select the input channel, change the input coupling for the channels, and turn the channel on or off. To display the outputs of a circuit, assign a Test Point to the nodes of interest.

Amplitude range controls

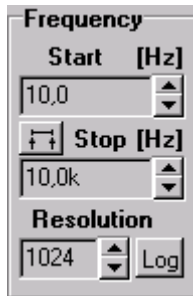


Increases/decreases the input amplitude range in a 1-2-5 step sequence. The amplitude units can be changed by pressing **dB** or **V**(olts).

The Auto button places the instrument in the autoranging mode. In this mode, the instrument automatically selects the best range to measure the applied input signal.

The Man button places the instrument in the manual ranging mode.

## Frequency range



The signal analyzer horizontal axis always represents frequency. You can adjust the beginning and end points by setting the start and stop frequencies. Alternatively, you can set a frequency to be centered on the display by setting the center frequency. The frequency span then is displayed symmetrically about the center frequency.

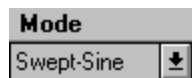


Pressing toggles between start-stop and center-span frequency functions.



Scales the horizontal axis divisions in logarithmic units when **Log** is selected. When **Lin** is selected, the horizontal divisions are linearly scaled.

## Measurement mode

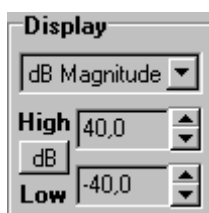


Allows you to select the measurement mode.

The available measurement modes are Swept-Sine, Amplitude spectrum, Amplitude spectral density, Power spectrum and Power spectral density.

In the swept-sine mode, the Function generator can generate linear or logarithmic sweeps with a selected sweep start frequency, stop frequency, and resolution.

## Display setup



Selects the type of analysis to be displayed. The available types are Lin magnitude, Log magnitude, dB magnitude, phase, Bode (gain and phase), Nyquist and Group Delay.

Specifies the vertical axis scale by adjusting its Low and High values.

## Control



Displays one of four measurement setup menus.

## To trigger the signal analyzer

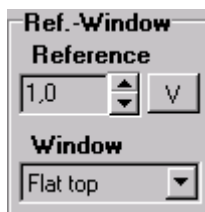


Allows you to select a trigger mode or source.

Changes the trigger level.

Selects either the rising or falling edge of the signal to trigger the signal analyzer.

## Reference and Window function setup



Allows the reference level to be changed. The reference level is the amplitude power or voltage represented by the 0 dB line on the display. Changing the value of the reference level changes the absolute amplitude level (in the selected amplitude units) of the 0 dB line.

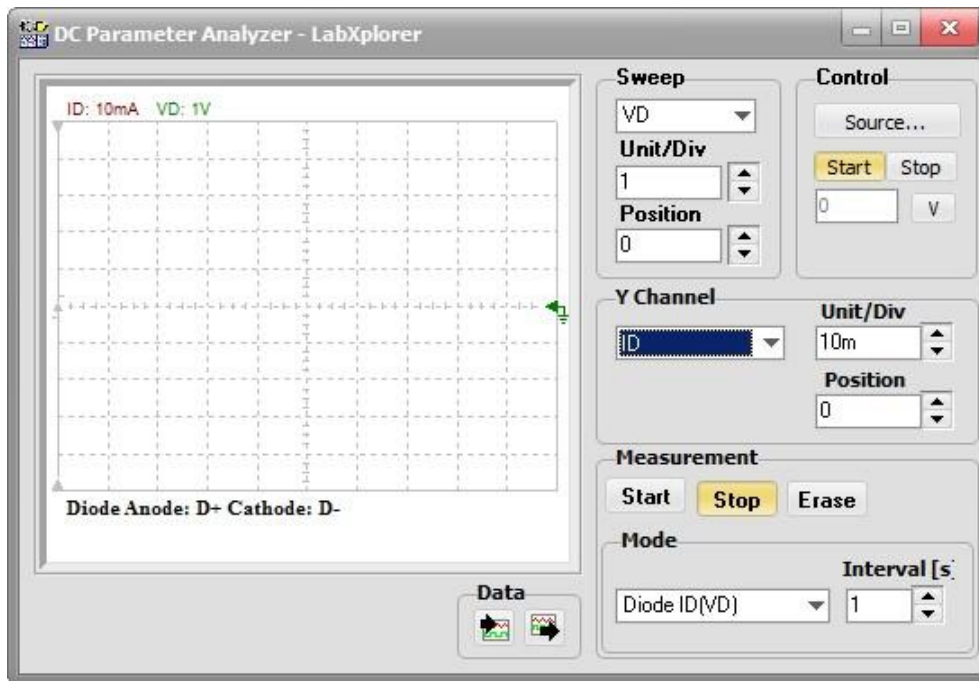
Selects the type of window function. The available types are Hanning, Flattop, and Uniform.

If we multiply our time record by a function that is zero at the ends of the time record and is large in the middle, we would concentrate the FFT on the middle of the time record. Such functions are called window functions because they force us to look at data through a narrow window.

## DC Parameter Analyzer

The DC Parameter Analyzer is the measuring instrument which can get semiconductor characteristics, like diode current-voltage, BJT input, transfer and output, MOS ID-VGS and ID(VGS, VDS). Transistors can be of N or P type.

The instrument controls multiple generator resources of LabXplorer such as DC Power Supply and Function Generator and records current and voltage data.



The first step of using this instrument is always to select the measurement mode. Then the Parameter Analyzer configures the terminals (D+, D–, B) for the selected semiconductor device. The process enables voltage/current sources and voltage/current meters within the box.

When you press the Start button, the analyzer forces the signal source(s) to step across a previously set range of interest, displaying the measured characteristic values.

### Connectors

- B Output terminal to the base (or gate) pin of the transistor for measuring its transfer characteristic.
- D+ Output terminal to the DUT (device under test) providing programmable voltage for measuring two terminals or two-port devices characteristics.
- D– Input terminal to measure the DUT current.

## To set up the Measurement Mode

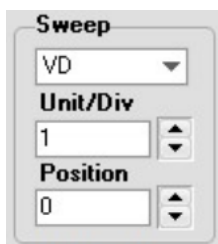


These controls allow you to select the Measurement Mode, change the input and output configuration of the DUT sockets (B, D+, D-) as well as the period the data acquisition. Under the coordinate system of the virtual instrument there are hints how to connect the DUT.

The Parameter Analyzer can change the output configuration of the Function Generator. Depending on the selected measurement mode, one or two output channel name(s) will be adjusted.

These channels will be controlled by the characteristic measurement and their new names will additionally contain the signal name as well as the output socket name. For example, if the 'NMOS IDS(VGS, VDS)' mode is selected then two channels, the 'VDS=D+(+V)' and 'VGS=B[V]' will appear. The combined names inform which output will generate which signal on which connector.

## Sweep



Select horizontal channel.

Changes the display scaling in a 1-2-4 step sequence.

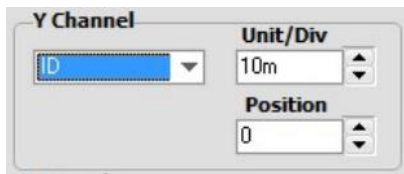
Positions the waveform horizontally.

## Control



The parameter analyzer horizontal axis (the swept channel) is always controlled by the function generator. You can view the beginning and end values by pressing the Start and Stop buttons. Clicking the Source... button you can set the start and stop values of the sweep values of the function generator.

## Vertical Control



Selects the input vertical channel.

Changes the vertical scaling in a 1-2-5 step sequence.

Positions the characteristic curve vertically.

## Storage Control

There are three storage keys to change the operating mode of the Analyzer.



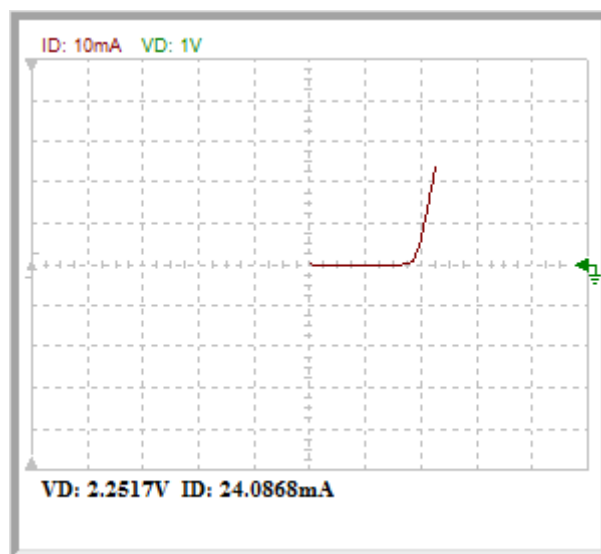
The analyzer acquires data and displays the characteristic curve(s).

Freezes the display.

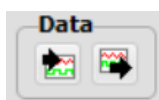
Clears the display.

## Display Area

In addition to displaying characteristic curves, the display provides range (unit/div) at the top and shows the currently measured values at the bottom.

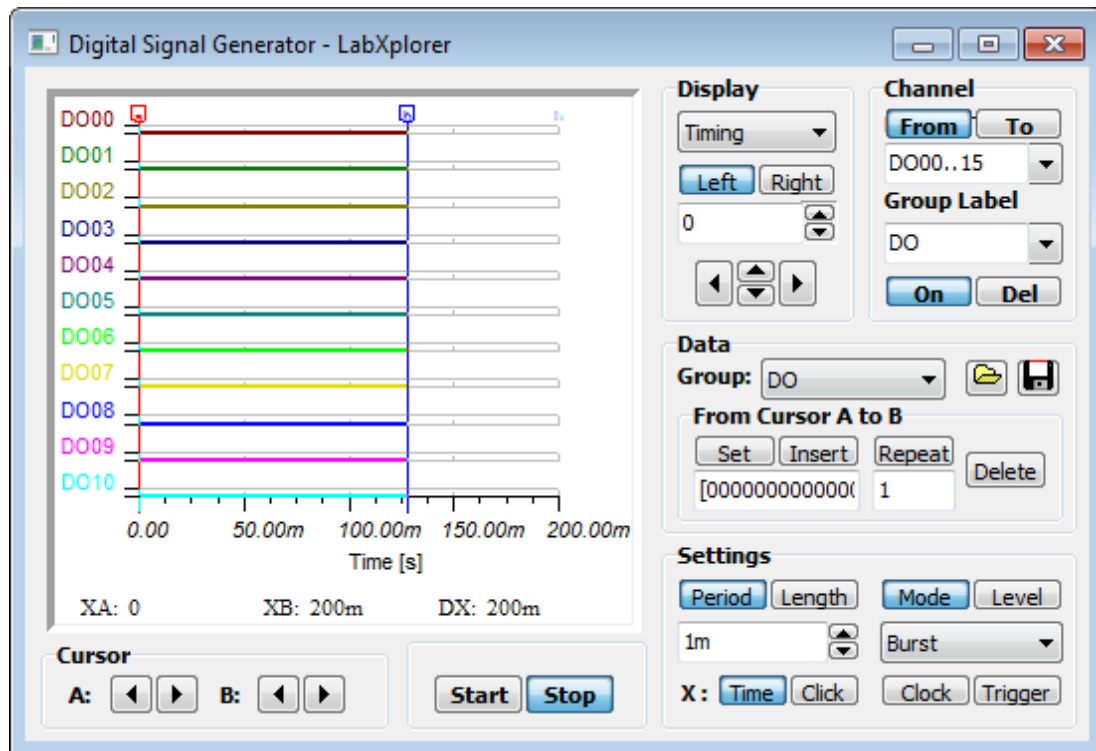


## Data Export



The analyzer imports from and exports to the characteristic curve(s) into the Diagram Window.

# Digital Signal Generator



## Display

Allows you to select the display mode and change the waveform display range. Zooming lets you change the waveform range in the display.



The only Timing Display mode available at present is the timing waveform.

Left: Position the left side of the desired zoom interval

Right: Position the right side of the desired zoom interval



Adjust the selected side of the desired zoom interval.



allows you to Shift the coordinate system left or right and scroll up or down.

## Channel

Under the channel options, you can combine signals into channel groups and provide them with labels. A group is simply a container for any number of signals arranged in any order. All signals must belong to a group even if there is only one signal in the group. Groups are typically created prior to signal assignments.

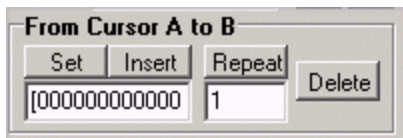
A dialog box titled "Channel". It contains two columns of dropdown menus labeled "From" and "To". The "From" dropdown shows "D000..15". Below these is a "Group Label" dropdown showing "D0". At the bottom are two buttons: "On" and "Del".

Allows you to define channels to form a group by specifying the first and last channels in the group.

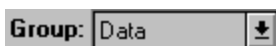
Allows you to assign a label to the group, or to select another group.

Turns the group on or off.

## Data entry and modification

A dialog box titled "From Cursor A to B". It has four buttons: "Set", "Insert", "Repeat", and "Delete". Below "Set" is a text box containing "00000000000000". Below "Repeat" is a text box containing "1".

You can Set, Insert, Repeat and Delete a pattern of bits between the position of the A and B cursors. You can specify pattern words by typing 1s and 0s in the box. The left-most bit applies to the uppermost signal. If you do not want to modify the value, type an X. If you use Repeat, give the number of cycles to be repeated.

A dropdown menu labeled "Group:" with "Data" selected and a downward arrow button.

Selects the channel group to be edited.



Loads or saves the data patterns.

## Settings

A settings dialog box with four sections: "Mode" and "Level" (each with a dropdown), "Step" (with a dropdown and a downward arrow button), and "Clock" and "Trigger" (each with a dropdown).

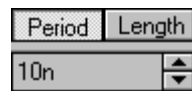
Mode allows you to select the Data output mode: individual steps, burst, or continuous.



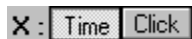
The Level setting enables the generator to handle different logic families, including TTL and CMOS.

The Trigger source may be the external or internal logic analyzer.

The Clock source may be the external or internal logic analyzer.



Specifies the clock period or recording length.



Switches the time scale between absolute time and clock period time.

## Cursors

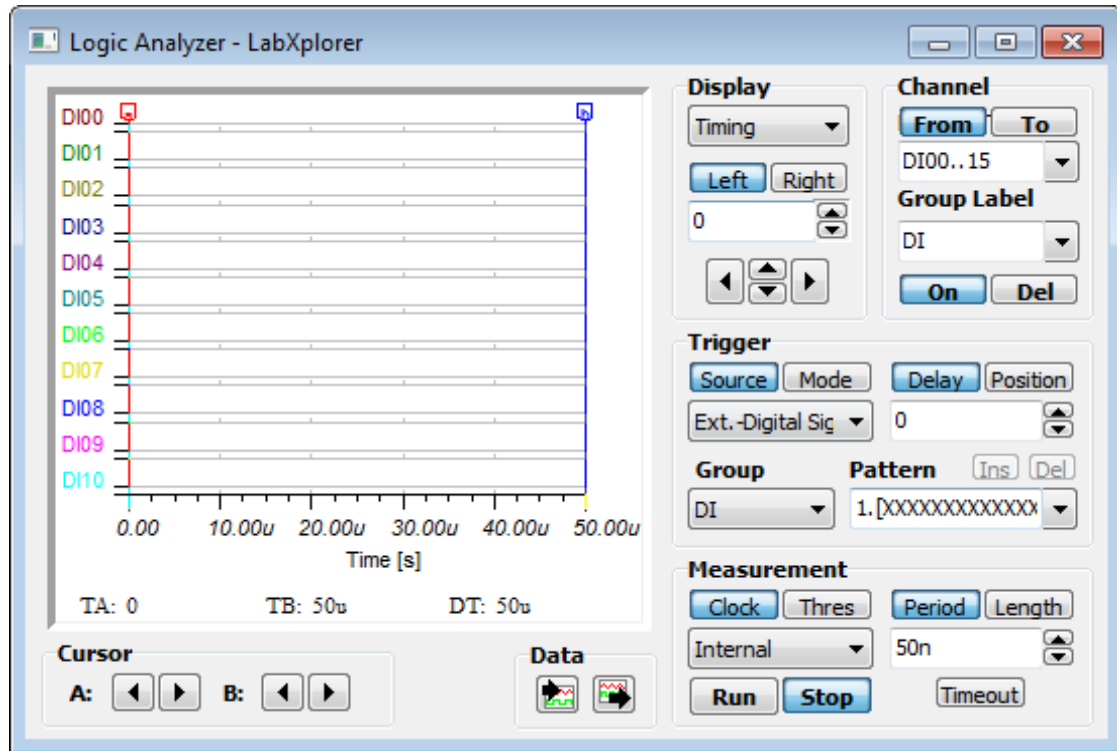
The Data Signal Generator provides two cursors which you can use to identify points where the signal value changes. You can also make time delta measurements between the two cursors. The time at the position of each cursor is always displayed on screen, as well as is the delta.



left and right cursor movement buttons

## Logic Analyzer

The logic analyzer is useful for the accurate timing examination of digital signals.



### Display

Allows you to select the display mode and change the waveform display range. Zooming lets you change the waveform range in the display.



The only Timing Display mode available at present is the timing waveform.

Left: Position the left side of the desired zoom interval

Right: Position the right side of the desired zoom interval



Adjust the selected side of the desired zoom interval



Shift the coordinate system left or right and scroll up or down.

### Channel

In the channel block, you can combine signals into channel groups and provide them with labels. A group is simply a container for any number of signals arranged in any order. All signals must belong to a group even if there is only one signal in the group. Groups are typically created prior to signal assignments. Digital trigger patterns can be associated with groups.

The image shows a software interface for managing channel groups. It has a title bar 'Channel'. Below it are two dropdown menus labeled 'From' and 'To', with 'DI00..15' selected in the 'From' menu. Below these is a 'Group Label' dropdown menu with 'DI' selected. At the bottom are two buttons: 'On' and 'Del'.

Allows you to select a group of channels by specifying the first and last channel items in the group

Allows you to assign a label to the group, or to select another group.

Turns the group on or off.

## Triggering the logic analyzer

One of the most powerful characteristics of the Logic Analyzer is its ability to trigger on a wide range of hardware conditions across multiple signals. In the Trigger pane, you can set up the conditions which will cause the logic analyzer to start acquiring data. You can construct trigger conditions ranging from simple hardware high or low conditions to code execution sequences.

The image shows a control panel for trigger settings. It has two tabs: 'Delay' and 'Position'. Below the 'Position' tab is a numeric input field containing the value '0' and a small up/down arrow button.

Trigger delay and position settings let you quickly specify the trigger position in memory.

The image shows a control panel for trigger source and mode. It has two tabs: 'Source' and 'Mode'. Below the 'Source' tab is a dropdown menu with 'Sequential' selected and a down arrow button.

Allows you to select the Trigger Mode and Trigger Source. When External Trigger Source is selected, the Digital Signal Generator itself is used, while Internal Trigger Source triggers off of events detected by the Logic Analyzer based on the pattern(s) you've defined.

When an Internal source is selected, you can select Conditional or Sequential trigger mode. Conditional mode uses one trigger pattern, but Sequential mode can use two or more trigger patterns occurring in sequence.

The image shows a control panel for selecting a group. It has a title bar 'Group' and a dropdown menu with 'Data' selected and a down arrow button.

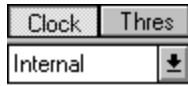
Selects the channel group to be edited.

The image shows a control panel for defining a trigger pattern. It has a title bar 'Pattern' and two buttons: 'Ins' and 'Del'. Below the title bar is a text input field containing '1.[00100100]' and a down arrow button.

The trigger pattern defines the point in time at which data acquisition is to be started. You can specify trigger pattern words by typing 1s and 0s in the box. If the level can be 0 or 1, type an X. You can insert a pattern before other pattern(s), and you can delete unnecessary pattern. Identical sequential trigger patterns are unified

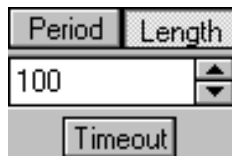
before being stored into the instrument trigger memory. For the number of words and the bit width of the trigger pattern, see the Specification.

## Measurement parameters (Acquisition Control)



Allows you to select a Threshold mode suitable for different logic families and voltages, including TTL and CMOS.

Allows you to choose an external or internal clock source. Currently, External is not available.



Period - Specifies the clock period for the sampling. Setting the correct clock period (it can be compared to the timebase setting of the common oscilloscope) is vital if the signal you capture is to be accurate.

A general rule is that the Analyzer's sampling rate should be a minimum of twice the frequency of the fastest signal that you are sampling. If the logic analyzer is not sampling fast enough relative to the system under test, information will be missed between sampling points.

The faster the sampling, the better the resolution, but you should also realize that increasing the sampling rate decreases your sampling time. Sampling time can be calculated as sample frame memory depth times the sampling period. For memory data, see Specification.

Length - Specifies the recording length in Clock Periods.

Timeout - If the programmed trigger events do not occur within the pre-defined time limit, the Timeout will activate and cause a trigger.

## Cursors

The Logic Analyzer provides two cursors which you can use to identify points where the signal value changes. You can also make time delta measurements between the two cursors. The time at the position of each cursor is always displayed on screen, as is the delta.



left and right cursor movement buttons





# Revision History

Location	Page
09/2017 changed from REV. A to REV. B	
Changes to specification of Multimeter/DC Parameter Analyzer .....	11
Added Device characteristic measurement .....	45
Added Generator matching.....	52
Changes to Multimeter.....	53
Changes to Variable DC supply.....	59
Changes to Function Generator.....	58
Added DC Parameter analyzer .....	68