Venable ESTiView
Software Version 2.0

Software and Hardware Manual

For ESTi Models

2161, 2165, 2185
2361, 2365, 2385

For Nexus Models

2761, 2765, 2785
2961, 2965, 2985

Venable Instruments
8656 SH 71 West, Bldg. E
Cuesta Centre
Austin, TX 78735

512-949-3100

www.venableinstruments.com

TABLE OF CONTENTS

TOPIC **PAGE**

Introduction

This manual describes the Venable ESTiView Software that is part of Venable's ESTi (Energy Storage Test instrument) and Nexus battery testing systems.

This manual is provided in printed form with the hardware test device, in electronic form in the software and in electronic form on the download site. The latest version of this manual can always be found on Venable's download site beside the current software installation package. The version in the current software installation can be reached using the menu item Help → Venable Manual.

The Setup portion of this manual covers both the hardware and software setup.

The Software works with a number of different hardware models of Venable ESTi and Nexus multichannel systems. The Software adapts to the differences between models by showing different Control dialogs and making minor changes in other dialogs.

System Overview

The Venable ESTiView Software, in combination with a supported Venable ESTi or Nexus, is a complete measurement and analysis system.

The hardware portion consists of one or more Venable ESTi/Nexus, which is used for performing various operations on a battery, including charge/discharge operations, collecting data and performing AC impedance measurements.

The Venable ESTi/Nexus is controlled by this software using a standard Ethernet connection.

The ESTiView software runs on any personal computer using Microsoft Windows 7, 8, 10 or 11, 32-bit or 64-bit.

The Venable ESTiView software is protected against illegal copying and use. In order to connect or operate an instrument, or view data files, a matching license for the instrument must be activated. This is done through the menu item: Help -> License….

Setting up the System

Software Installation:

1. Close all your programs and insert the Venable installation disk into your CD-ROM drives. The CD should auto-run and the installation window should open up, otherwise browse the CD with Windows Explorer and double-click on VenableSetup.exe. Follow the prompts to allow the installer to install the Venable Software to your PC.

2. Make sure you have your software serial number at hand and follow the instructions.

   The Venable software is protected against illegal copying and use. In order to connect to instruments and modify data files, each type of device must be activated by the Licensing Wizard built in to the Venable software. From the software main menu, the wizard is accessible from Help -> License….

   Manuals for the Venable software, “Venable Software Manual.pdf”, and the Licensing Software, “LicenseManual.pdf”, are included in the installation folder. These manuals are in Adobe portable document format (pdf) and are compatible with Adobe Acrobat Reader. From the Venable software program you can open the manuals using the menu items “Help -> Venable Manual” and “Help -> License Manual”.

   If you need to uninstall the software for any reason, use Add/Remove Programs and choose the uninstall option.

3. Set PC Sleep mode to “Never” in Power & Sleep settings.

4. **If you will be running a test for more than a couple of days**, you need to disable the Windows Update service to insure that automatic updates do not interfere with your test, since Windows can be quite forceful in rebooting your PC. Do windows updates manually in between tests to insure Windows security is kept up to date. Also, go to Windows 10 Settings>Accounts>Sign-in Options and there should be an option that says "Require Sign-In". Click the drop box below it and select “Never” to insure that Windows does not log you off and interfere with your testing.

Hardware Setup:

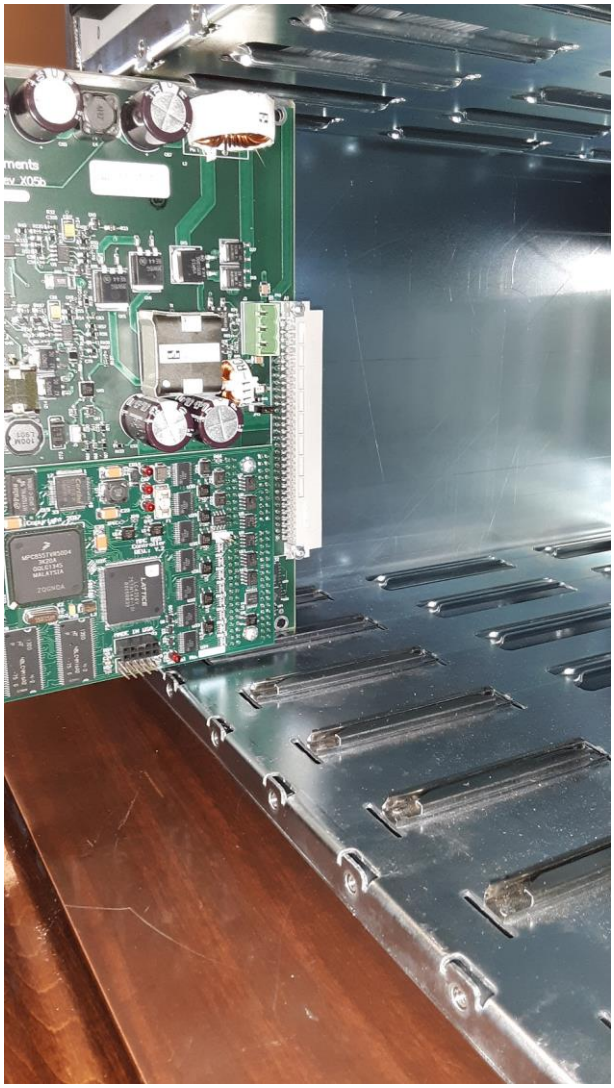
ESTi Unpacking and Setup:

Remove instrument from shipping box and place on location to be used. It is recommended to save the shipping box for future recalibration shipments.

Connect AC power cable to instrument and plug into AC outlet

Nexus Unpacking and Setup:

Remove cardcage from shipping box and place on location to be used.
Remove each channel from shipping box and slide into slot in cardcage according to slot labeled for each channel. Channel slot assignments can be changed later, as needed.



If you purchased a unit with less than 10 functional channel boards, the following steps are required:

1. Slots 0, 1 and 2 must be populated with functional channels. These slots manage the 3 cooling fans on the Nexus.
2. You will need to insert the cover plate that has the “dummy board” extender into the very next slot after the last populated channel board slot. The remaining cover plates are installed on the remaining open slots after the “dummy board” extender plate. This is necessary to insure proper airflow and cooling of the Nexus system.

NOTE: *New units have tight channel grounding springs and may require significant force to slide board into slot. It is best to push primarily on channel faceplate below main socket to apply force.*

Rotate fasteners clockwise to tighten channels into cardcage slots. It is recommended to save channel shipping boxes for future recalibration shipments.

For multiple Nexus setups, alphabetic stickers are provided to identify each Nexus chassis. These stickers are applied just to the left of the power button on the front of the Nexus.
Connect AC power cable to instrument and plug into AC outlet.

Communication of the Venable software to the instruments is achieved using a standard Ethernet connection setup. Venable **strongly** recommends that connection of PC to the instruments be done using a dedicated Ethernet bus separate from the Ethernet connection used to network the PC to an office environment, otherwise unreliable

control and operation of the instruments could occur. This can be done by adding a second Ethernet interface to your PC.

Disclaimer: Direct operation of an ESTi or Nexus, via the global internet from a PC at a remote location outside of the facility containing your ESTi or Nexus, is not currently supported due to potential for data packet corruption. This method of operation is at your own risk.

Attaching PC to a single ESTi:

In most cases, an Ethernet cable can be attached directly from the PC to the instrument. There is a possibility that the PC Ethernet hardware may not support automatic crosslinking, in which case, you will need to use a crosslinking (null modem) cable instead.

Attaching PC to multiple ESTi:

A standard multiport Ethernet switch will need to be obtained. Attach the Ethernet cable from the PC to one of the ports. Attach the instruments to the remaining ports on the switch.

Attaching PC to a single Nexus:

An Ethernet cable can be attached directly from the PC to the back of the Nexus.

Attaching PC to a multiple Nexus:

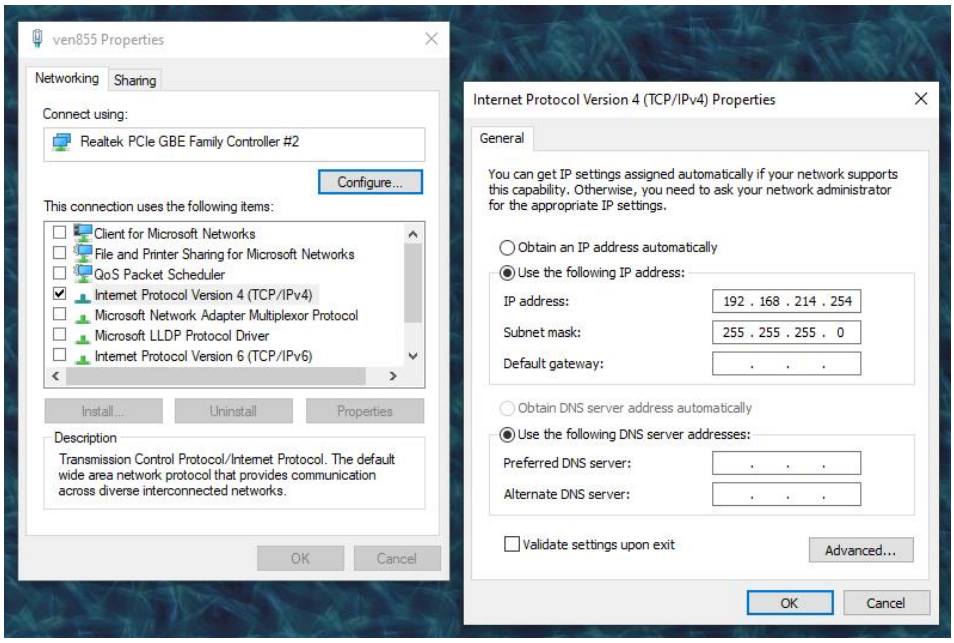
A standard 1 GB multiport Ethernet switch is recommended. Attach the Ethernet cable from the PC to one of the ports. Attach each Nexus to the remaining ports on the switch.
It is also possible to connect without a separate switch. An Ethernet cable can be attached directly from the PC to the back of the first Nexus. You can then daisy-chain the secondary ethernet ports on the back of the Nexus across Nexus units.

Ethernet IP Address Configuration:

The ESTi comes initialized to the default IP address of 192.168.214.214.

Each Nexus comes with boards initialized to default IP addresses of 192.168.214.214 thru 192.168.214.223 and are initially assigned in the order of slot assignments. IP addresses and slot assignments can be changed later, as needed.

In order to establish the initial connection of the ESTiView software to the instrument, the Ethernet interface in the PC will need to be configured to the IP address of 192.168.214.254, subnet mask 255.255.255.0. In Windows, open up the “Control Panel” application and select the “Network and Internet” category. Select “Network and Sharing Center”, then select “Change adapter settings”. Double click on the Ethernet adapter that will be used for the connection to the instrument. This will open up the properties window for that Ethernet port. Make sure that “Internet Protocol 4 (TCP/IPv4)” is check marked and click on the “Properties” button. This will open the window to change the IP address properties of the PC. Configure this window as shown in the figure below.

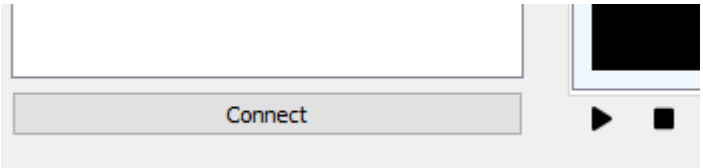


After an initial connection is established with the instrument, the ESTiView software can be used to reconfigure to a different instrument IP address and PC IP address. However, the instrument and the PC must be kept in the same subnet space. If using multiple PCs to operate groups of ESTi or Nexus instruments, each PC will require a unique IP address.

1. Turn on the ESTi or Nexus Instruments. Wait for the instrument to complete startup; the green “Status OK” LED on front of instrument will illuminate when startup has completed.

2. Open the Venable software. Select the Actions menu in the Venable Software and click on the “Connect” menu item. This will display the connection dialog. Enter the IP address of the instrument you want to connect to (for ESTi first time use, this will be the default value of 192.168.214.214) and click on the connect button. Instrument Port value should remain unchanged with the default value. When the software has successfully connected to the instrument, the Status display box will show a valid connection.

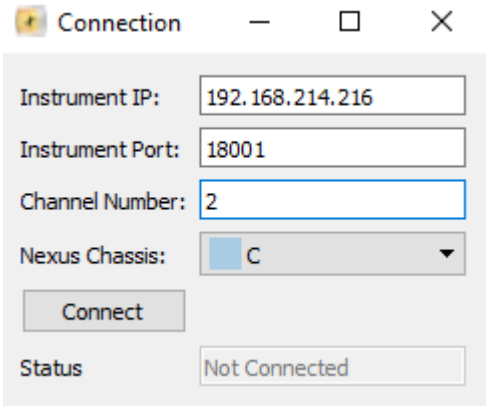
Connect Button



This button in the lower left section of the application window can be used to establish a connection with a particular instrument.

Clicking on this button will bring up a connection dialog box. Each instrument that is connected onto the Ethernet network must have a unique IP address assigned to it. See section for the Utilities Tab for details on setting an instrument to a new IP address.

Change the IP address to the address you would like to make a connection to. Then set the Channel number field to value that you would like to assign for this instrument. Each connection must have a unique value. Valid values are 0-15. Instrument Port value should remain set to the default value.

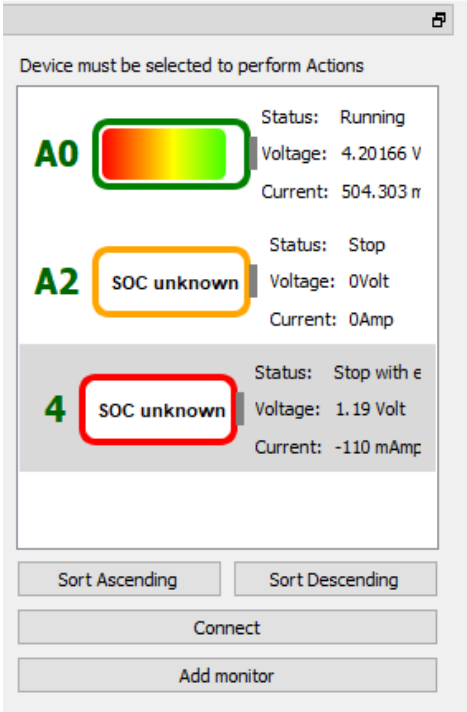


For multiple Nexus setups, a chassis letter, as designated by the stickers applied to each Nexus chassis outlined in the Nexus hardware setup section, can be specified for each Nexus to manage multiple chassis.

If you have more than 16 single channel ESTi units, the chassis selection can also be utilized to increase access to additional channels.

Click on “Connect” to connect the instrument to the software. If the connection is successful, it will appear in the connection list.

Connection List



This window shows active connections that have been made to devices under test. By default, it resides on the left side of the application window, but is moveable and can be undocked and moved using the bar at the top. For each device, the window shows 4 pieces of information. The status is shown, indicating whether a test is running or stopped. The active voltage and current levels are shown. Also the user assigned channel number and chassis letter is displayed in green on the left side of the battery symbol.
The battery icon has three border colors to indicate the battery status; orange color to show that no test is running on selected battery, green color, to show that the test is running, and red color to show that the battery is stopped with error.

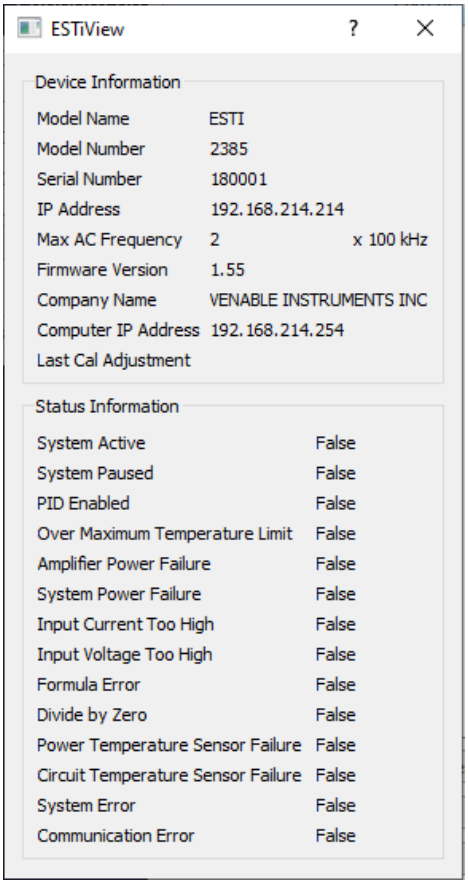
The label "SOC Unknown" appearing on the battery icon indicates that the battery's charge level (State of Charge, SOC) cannot be identified for the current test. Once the SOC is successfully calculated, this label will disappear, and the battery icon will show varying levels of charge.

The "Sort Ascending" and "Sort Descending" buttons allow you to arrange the channels in ascending or descending order, respectively.

Left clicking on a particular device will show the design currently associated with it in the Design tab. It will also set the focus of operations for the Sensors and Utilities tabs and the Capture Waveform dialog. Right click on a device will bring up a sub-menu with 2 options.

The Disconnect option will disconnect the device from the application and remove it from the list.

The Info option will bring up a dialog box that displays information about the selected instrument. There are two sections in the window: The top section shows information about the instrument. This information will include the serial number of the instrument, Ethernet IP address settings and firmware version information.



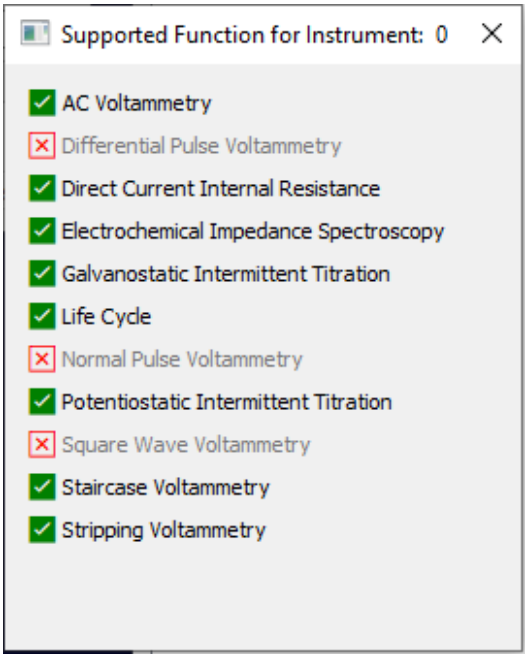
The bottom section will display status information of the instrument and some error information regarding the user defined test scenarios.

Some errors in a defined test scenario such, as a general error in a variable/formula definition, or an attempt to divide by zero in a variable/formula definition, will be recorded by the instrument and displayed in the status information window.

Also, any failure in power with the instrument, or with the main power amplifier driving the output of the instrument, will be recorded by the instrument and displayed in the status information window.

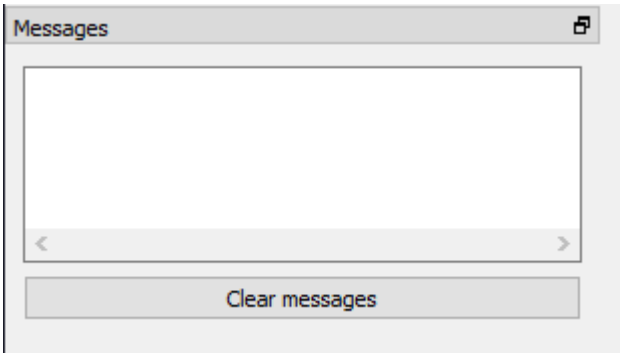
Support Function

This window displays the functions and tests that are supported by the instrument. Functions/tests that are available for use are indicated with a green check mark, while those that are not supported will appear grayed out and are denoted by a red cross.



Message Box

This section shows error and warning messages from the instrument and the ESTiView software.



Design Tab

Node Editor

This window is used to design new test scenarios. There are three main sections in this window, the icon selection bar along the top, the node layout and connection editor on the left and the node parameter editor on the right.

The icon selection bar

The icon selection bar is a drag and drop list of test types available to perform using the instrument. Use the mouse cursor to hover over any icon in order to display what type of control function that particular icon is.

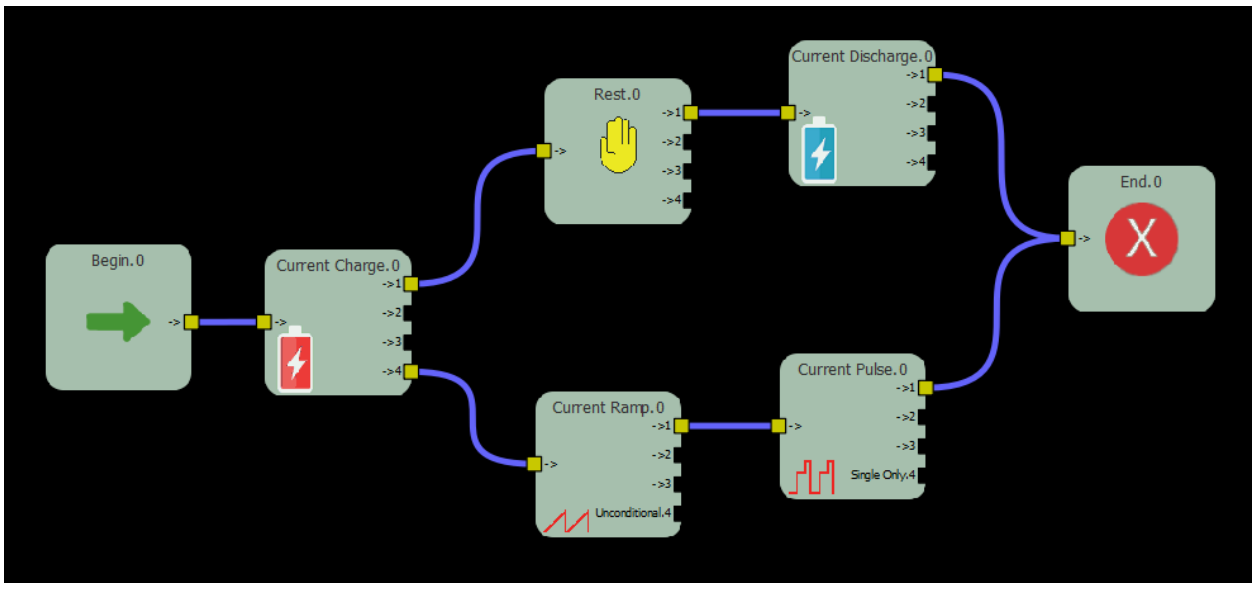
To drag and drop an item into the node layout and connection editor block, move the cursor over the icon and left click on the icon to select it. Then move the cursor into the node editor window to the position you want to place the icon. Left click again to place the icon into the editor. You do not need to hold the left mouse button down while dragging the icon into the editor.

Shortcuts

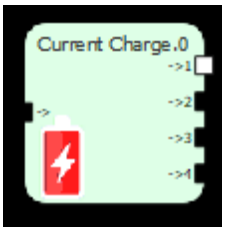
There are four shortcuts on the selection bar: The disk icon can be used to save your test definition. The folder icon can be used to Load a saved test definition. The copy can be used to copy the entire test into clipboard. The paste icon can be used to paste the recent copied test into the layout.

The node layout and connection editor

After placing icons into the editor window, you can move the icons around in the workspace by left clicking on the icon and holding the left mouse button down. To place the icon in the new location, release the left mouse button.



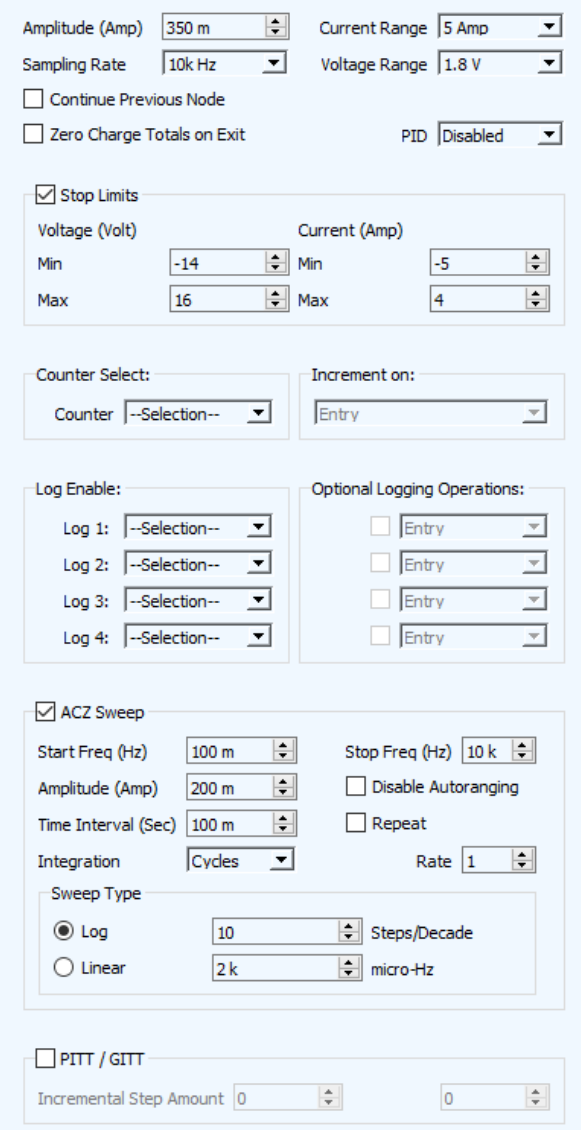
Each icon has one input connection (left side of icon) and 4 output connection (right side of icon) points for linking transition flow between icons. In order to link two icons, place your cursor to point to any output connection (the connection point will be highlighted with a white block, as shown), left click and hold while moving the cursor to any input connection. Release the left mouse button on the input connection chosen. A line should now exist between the two icons linking them together.



The viewable space in the editor is larger than what is visible at any time. It provides the ability to zoom in or out in the viewable area, and to move the viewable area of the window to a different section of the test layout. In order to zoom in/out, move the cursor to any location within the editor and then use the scroll wheel on your mouse to zoom the viewable area in or out. To move the viewable area of the window to a different section of the test layout, move the cursor to any location within the editor, left click and hold the button while moving the mouse. Releasing the mouse button will set the focus of the viewable area to the new location.

To edit the control parameters of any particular node, position the cursor over the icon in the layout and connection editor and double click the left mouse button to select that node and display the control parameters for that node in the node parameter editor on the right side.

The node parameter editor



The node parameter editor is used to configure various aspects of the control node selected. The node ID value that the software assigns to the selected node is shown at the top of this box. The node editor consists of several sections.

The top section contains the elements to set channel measurement ranges and set the amplitude levels of the output. Current Range adjusts both input measurement and output generation ranges together. Voltage Range only affects input measurement range. Output Voltage Range is fixed to 18V.

The next section consists of Stop Limits associated with the selected node.

The next section consists of enabling event counters that were defined in the Formulas tab. Enabled counters can be set to increment on node entry or node exit.

The next section consists of Log enable and selection parameters for data collection purposes for the selected control node.

The next section contains additional measurements or capabilities that are specific to the control type being performed. Varieties of this section are explained in further detail below.

ACZ Sweep ***(Requires supported model)***

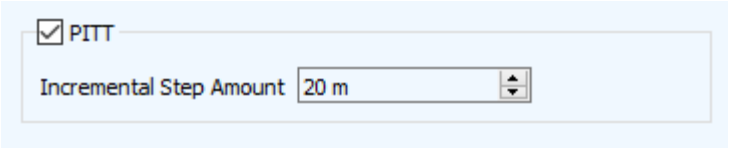
(This section is associated with DC charge/discharge nodes.)
An AC impedance sweep can be enabled during execution of the charge/discharge control by clicking on the checkmark box for ACZ Sweep. Start and Stop frequencies can be set to sweep either up or down in frequency. Amplitude of AC waveform and integration methods and times are also set up in this box. Care should be taken to insure that “ACZ Sweep Complete” is used as part of the exit condition criteria to insure that the control does not attempt to exit before a frequency sweep has completed. “The ACZ Sweep Complete” exit criteria is a TRUE (1) or FALSE (0)

operation and should be normally setup in your condition as “ACZ Sweep Complete > 0”. This function is available in both potentiostatic (voltage) controls and galvanostatic (current) controls.

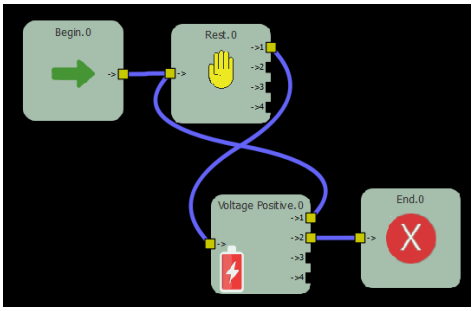
PITT / GITT

Potentiostatic / Galvanostatic Intermittent Titration Technique

(This section is associated with Voltage and Current charge/discharge nodes.)



This is used to enable incremental steps in amplitude to support this testing technique. A charge/discharge and rest loop (see figure) must be defined in the node editor for proper operation of this technique. In potentiostatic mode, the rest voltage will automatically carry over as the output amplitude into the first entry of the charge/discharge operation. It may also be required to enable PID feedback control to maintain constant output level in potentiostatic mode.



AC Voltammetry ***(Requires supported model)***

(This section is associated with Ramp nodes.)

An AC voltammetry operation can be enabled during execution of a ramp node function. This function performs a single frequency impedance measurement at regular repeating intervals during the ramp function. The injection frequency and amplitude are specified by the user, as well as the integration method and time for the measurement. This function is available in both potentiostatic (voltage) controls and galvanostatic (current) controls.

Pulse Editor

(This section is associated with Pulse nodes.)

This section is used to create a pulse waveform definition. Right click inside the definition block to bring up the “insert” button and then left click on the button to insert a step into the definition. Move the mouse cursor into the Time field and perform a single left click to select the field. Then enter the time value into the field and press enter to save the value in the field. Modifying the Amplitude field is done the same way as the Time field. A visual representation of the defined waveform is displayed below the definition box to provide a visual verification of the designed waveform.

NOTE**:** *The Time field may be automatically restricted or adjusted based on the setting of the Sampling frequency*. *The Amplitude field may be automatically restricted or adjusted based on the Range settings*.

Node Types

Begin: Marks the starting point of any test scenario. The Start Test menu option will initiate testing from this point. The Begin must be connected to one node only. Test initiation will immediately transition into this next node when a start is initiated. A begin node is automatically added to the node layout editor whenever a new test is created.

End: Marks the completion point of any test scenario. A test can have multiple completion points. Adding several End nodes is allowed for test design. Also, multiple node outputs can be connected to a single End node.

Rest: This control is used to pause testing and disengage the output relay on the instrument in order to fully disconnect the battery from the instrument to prevent loading effects. Only the Voltage sense probes are still connected to device under test to monitor voltage levels.

Current Charge: Performs a DC current charge on the device under test at an amperage level specified by the user. AC impedance sweeps on supported models are also available with this function.

Current Discharge: Performs a DC current discharge on the device under test at an amperage level specified by the user. AC impedance sweeps on supported models are also available with this function.

Voltage Positive: Applies a constant positive DC voltage to the device under test. Charging or discharging depends on the active voltage level of the device. AC impedance sweeps on supported models are also available with this function.

Voltage Negative: Applies a constant negative DC voltage to the device under test. Charging or discharging depends on the active voltage level of the device. AC impedance sweeps on supported models are also available with this function.

Current Ramp: Performs a linear current change from a defined amperage level to a second defined amperage level. Time duration for the linear change is defined by the user. Current Ramp nodes can be added back-to-back to create other waveforms such as saw tooth and triangle waveforms. The maximum attainable frequency of such waveforms is around 1 kHz. The AC voltammetry function on supported models is also available in conjunction with the current ramp function.

Voltage Ramp: Performs a linear voltage change from a defined voltage level to a second defined voltage level. Time duration for the linear change is defined by the user. Voltage Ramp nodes can be added back-to-back to create other waveforms such as saw tooth and triangle waveforms. The maximum attainable frequency of such waveforms is around 1 kHz. The AC voltammetry function on supported models is also available in conjunction with the voltage ramp function.

Current Pulse: Injects a user defined pulse train of up to 256 transitions of various amplitudes and time durations for each pulse. Output frequencies of up to 100 kHz are possible with this function. The pulse train will repeat until an exit condition is met. A “Single Only” option is available to inject only a single sequence of the pulse train and immediately exit to the next control node.

Voltage Pulse: See Current Pulse above.

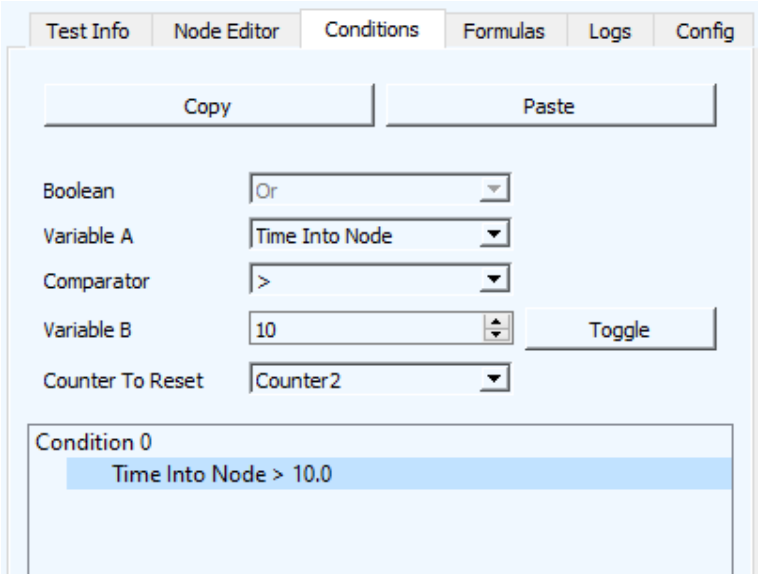
Power Charge: Perform a charge on the device under test at a constant power level specified by the user.

Power Discharge: Perform a discharge on the device under test at a constant power level specified by the user.

Load: Sets the instrument to act as an electronic load at the resistance (ohms) level specified by the user. This function acts as a discharge function only.

Conditions (editor)

Double left mouse click on a connection link in order to select that link for editing. The link will be highlighted yellow and the focus of the editing window on the right will change to bring up the exit condition criteria for the selected link. The condition editor contains three main parts. The copy/paste buttons are at the top, the variable and criteria selection is in the middle, and the condition listing for the node exit criteria is at the bottom. This window will initially be blank after creating a new link between nodes. Right click in the window to bring up a button to add an exit condition and then left click on the button to add a new condition. Several exit conditions can be added for one link.



Each condition listing must then have one or more expressions added to the condition to create one exit criteria. To add an expression, move the cursor over the desired condition and right click to bring up the button to add an expression. Left click on “Add Expression” and a new expression equation will be added below the condition. The criteria for the expression are modified by left clicking on the expression.

Use the criteria selection boxes above the listing box to change the expression values. In order to create an expression, a user will select a variable, either a default variable or a user defined variable for “Variable A”, to compare against either another variable or against a static value in “Variable B”. Variable B has a toggle button to transition between variable selection and static value editing.

When adding 2 or more expressions to a condition, the expressions must be related together using the Boolean operator. In a situation of 2 expressions, this determines whether only one or the other needs to be true (Or) for the exit to occur, or whether both need to be true (And) for the exit to occur. The “And” operator can also be used to separate multiple groups of “Or” criteria. This allows the user to create complex exit conditions for a particular node.

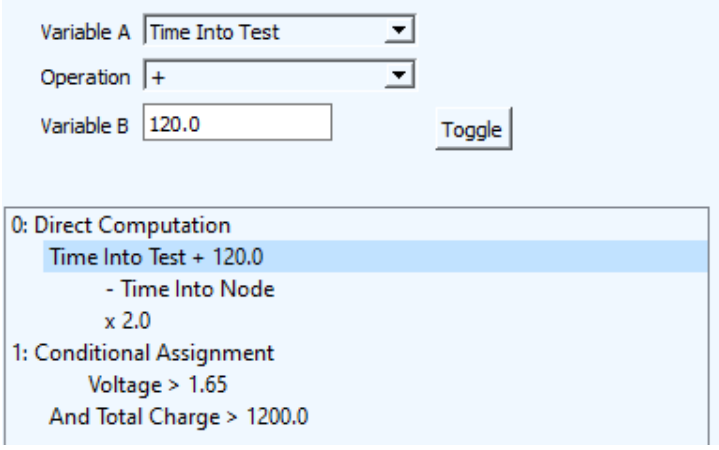
When more than one condition is added to a condition link, the control node will exit based on either condition being true. Each condition and expression that is added will incur an increase to the instrument processing utilization. When doing pulse or ramp functions at higher sampling rates, less processing time may be available and large condition definitions may not be possible.

The condition editor has a “Counter to Reset” drop box which contains a list of all the available event counters. This drop box allows the user to select an event counter to be reset when either condition is true.

The copy/paste buttons allow users to copy the exit condition criteria for the selected link and paste it into another exit condition criteria link for the same test. This feature will save time when there are many links that have the same exit condition, especially when the exit condition criteria are complicated.

Formulas (Variable creation / editor)

The primary of formulas is to create user defined variables for use in condition operations or for use in formulas to create other user defined variables. Complex equations can be created using multiple variables together. When creating complex equations using multiple variables, the order of variables and use in formulas is important to insure that all intermediate calculations are performed on the same processing cycle as the final variable calculation.



The formula editor contains two main parts. The variable and criteria selection is at the top and the formula listing is in the box below it. To add a new variable

with an associated formula, right click in the listing box. A menu of 3 choices to create, Direct Computation, Conditional Assignment or Event Counter, will appear. Left click on the choice to create it. Each of the two choices is explained in more detail below. Move the mouse over the new variable method created and left click on it to select the newly created variable with its associated method. In the editing boxes above, you need to select whether this new variable is to be defined as an integer or as a floating point value. Integer values are more efficient for processing utilization and should be preferred over floating point values whenever possible. In the Name box below it, you can specify your own name to identify the variable by simply selecting the default text and replacing it with new text.

Move the mouse over the new variable method created and right click on it to bring up the menu to add a new expression to the method, then left click on it to add it to the list. Left click on the expression to select it and bring up the expression editor above the listing box. Select a variable from Variable box A, and then select an operation to perform with the variable on either another variable from Variable B or an editable value. The toggle button to the right of Variable B will switch between variable selection and editable values. You can add up to 8 expressions for each Direct Computation or Conditional Assignment. Each formula and variable that is added will incur an increase to the instrument processing utilization. When doing pulse or ramp functions at higher sampling rates, less processing time may be available and larger definitions may not be possible.

There are two distinct methods to create a new user defined variable; Direct Computation and Conditional assignment.

Direct Computation:

This performs an arithmetic operation on a combination of variables and/or static values. The operations that can be performed are plus, minus, multiply and divide. If the variable is being defined as an integer, the list of operations will include the binary operations of “AND” and “OR”.
When adding 2 or more expressions, the subsequent operations will be performed between the result of the preceding expression and the Variable B selection of the new expression. See appendix A for examples of Direct Computations.

Conditional Assignment:

This performs a “compare and set” operation by comparing a chosen variable in Variable A, against a combination of other variables or static values. The new variable will be set to the value of the variable in the first instance of Variable A when the entire expression set has evaluated to TRUE. The new variable created must be of the same type, integer or float, as the variable selected in the first instance of Variable A, otherwise a computation error will occur. The operations that can be performed are ‘less than’, ‘less than or equal to’, ‘greater than’, ‘greater than or equal to’. Multiple expressions in a conditional assignment are always ANDed together for compound expression evaluation. See appendix A for examples of Conditional Assignments.

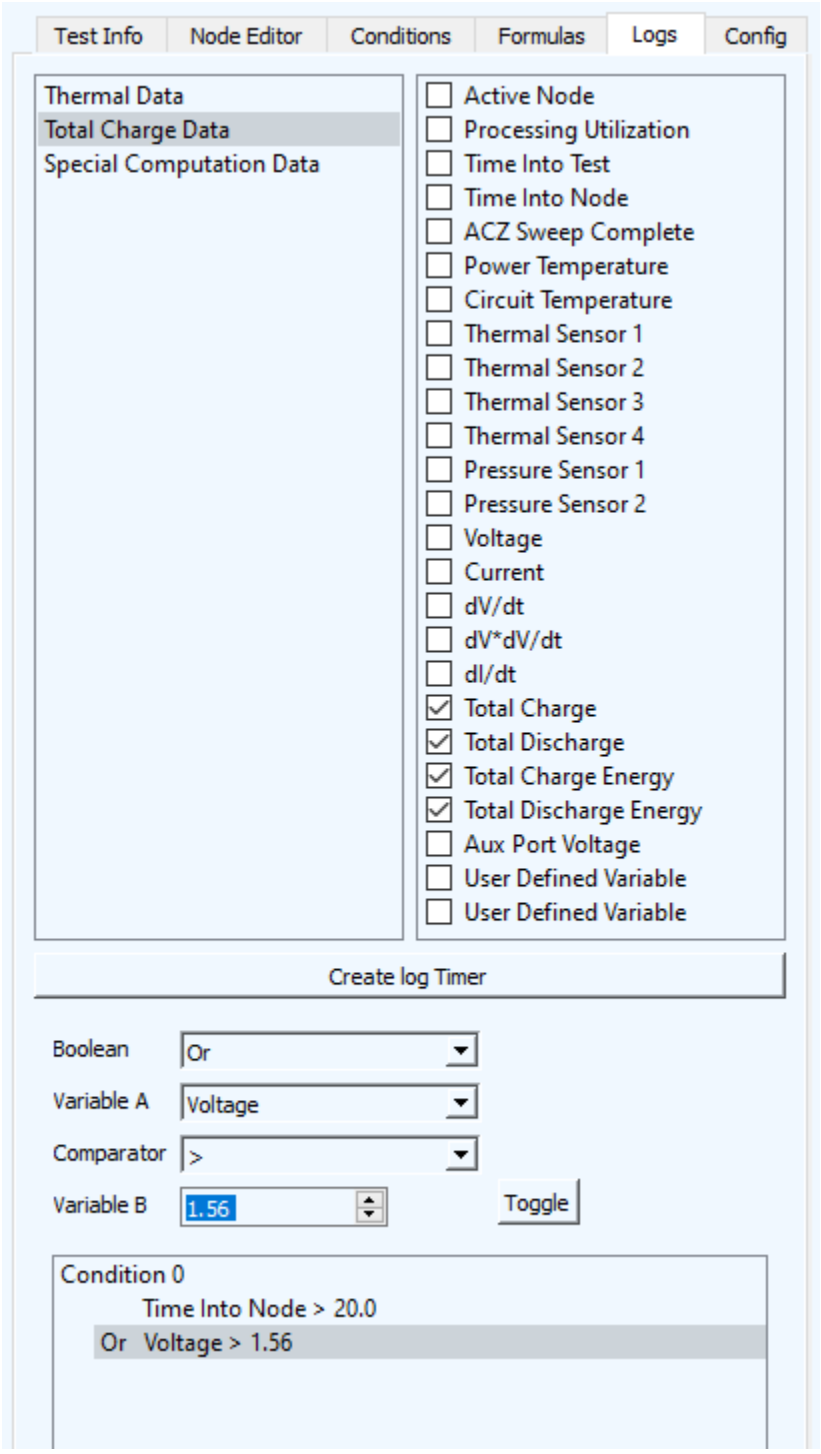
Event Counter:

This creates event driven counters that can be used to keep track of the number of cycles that have occurred in a loop. There are no expression definitions for event counters. Counters can be configured in the Node Editor.

Logs

Logs are the primary method of data collection provided by the instrument and software. The logs allow the user to select one or more variables, default or user-defined, to capture and store into a file for future analysis. The logs tab is split up into three sections. The list of user defined logs is created and displayed in the top left box. The list of variables to select from is displayed in the top right box. At the bottom of the window is the condition editor used to define conditions when a log of data should be recorded.

To create a new log, move the cursor into the top left box and right click to display the menu. Left click on “Add Log” to create a new log definition in the list. Left click once on the log to display the variable selection list for that particular log in the list on the right side. Variables to log are selected by setting a check mark next to the variable. Log names can be changed with your own names by double left clicking on the log definition on left side. This will highlight the text and allow you to change it with your own text.



Unselect the log definition to save the new name.
A logging of data can be set up to occur in one of three ways. 1) On entry into a control node, 2) On exit out of a control node, or 3) Based on a condition event occurring. Log definitions are available to all control nodes and are enabled for each control node independently. Once a log definition has been created, the log can be enabled in the node editor for a particular control node. Each node allows activation of up to 4 log definitions. In order to enable a log, select the control node of interest in the node editor by double clicking on it to bring up the node editor screen on the right side. Selecting a log from the pulldown list of logs in the Log Enable section, will activate the log for that control node and will generate a data log based on the log conditions that were set up for that log. A log can also be generated on “entry into” or “exit from” a control node by check marking the check box next to the log selection and then selecting the choice of entry or exit in the pulldown menu next to the check box. Entry/Exit settings can be used in conjunction with condition-based logs in a particular node.

To delete a Log definition, move your cursor over the definition to delete and right click on it. A menu will appear. Left click on “Delete Log” to remove the log definition.

Transmit Log (Manual Request): When you right click on a log definition, the menu will also show an option to transmit a Log on request. This can be used to test your log definition. Received log data will be stored to a file on the PC. In order to use this function, you must have created a log definition, connected to an active instrument and have loaded your test definition into the instrument.

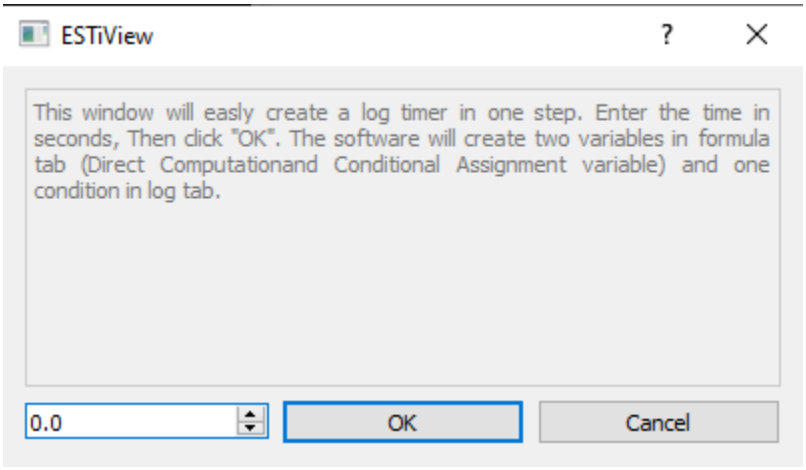
Conditions (editor):

Each log definition can have its own set of logging conditions set up for it. To create a condition, select the log in the top left list. Move the cursor down to the condition listing at the bottom. Right click in the window to bring up a button to add a condition and then left click on the button to add a new condition. The method of adding and defining expressions to the conditions and adding additional conditions is identical to how it is done for node conditions. Reference the conditions (editor) section for details and restrictions in defining conditions for data logging.

Logs at timed intervals:

It is often necessary to collect data at regular timed intervals. See Appendix A for an example of how to set up your formulas and log conditions in order to accomplish this function.

To create a timer interval to log data, click the “Create Log Timer” button. A small dialog will appear which allows the user to enter the time interval in seconds. Enter the time and click “OK”. The software will create two user-defined variables (direct computation and conditional assignment) in the formula tab and one log condition in the log tab.



Config

The config tab allows you to configure the instrument itself at an overall test level. These settings apply to the entire test, not for any particular control node.

Monitoring Interval: The instrument can transmit the entire variable data set at regular intervals for general purpose monitoring in the Monitor tab window. The interval setting has a resolution of 0.1 seconds. The interval can also be set to 0 to turn off generation and transmission of this data completely. Lower interval settings, other than 0, will incur a performance hit on Ethernet bandwidth availability and on instrument processing utilization.

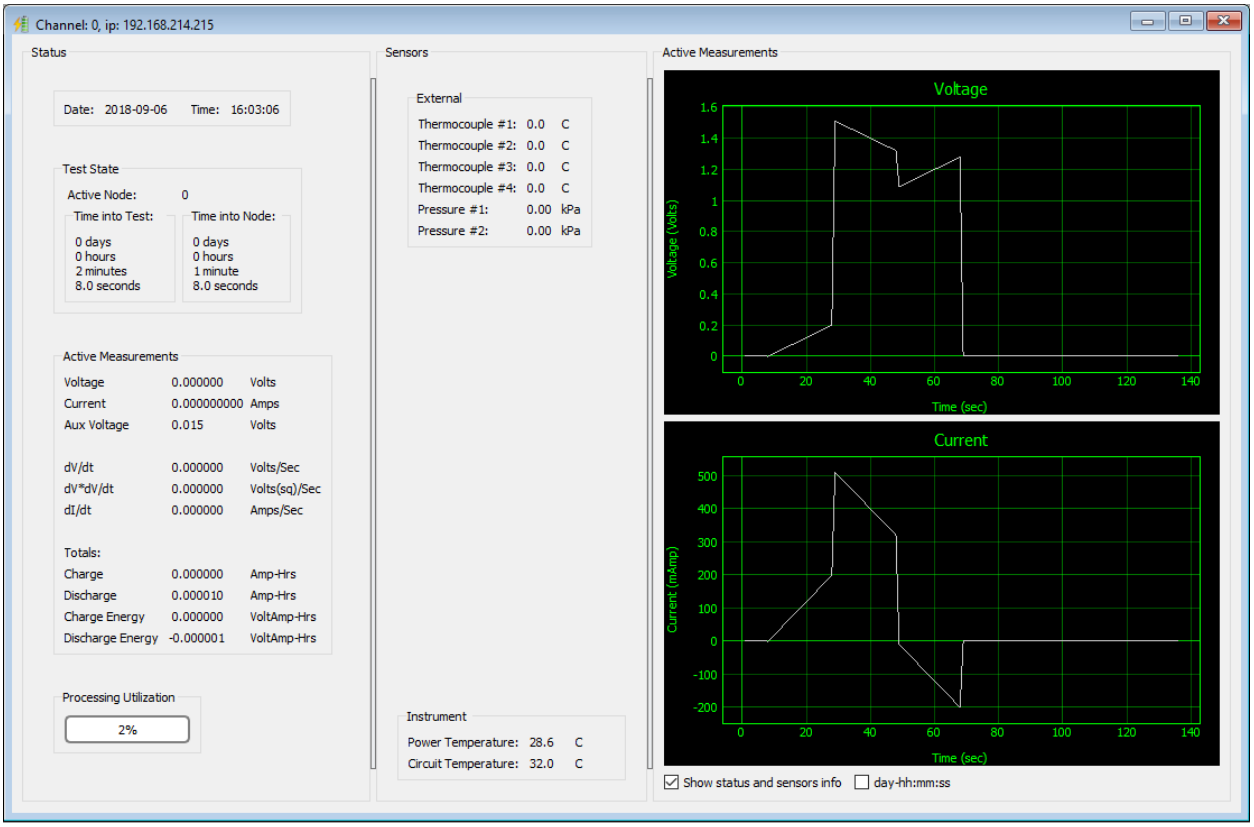
Shutoff Voltage/Current: The purposes of these settings are to prevent over-voltage or over-currents from occurring that can damage a device under test. These settings can be used to configure the instrument to stop all testing if defined limits are exceeded.

Aux Port Range: When using the auxiliary port as a secondary voltage sensor, this setting will set the port to the applicable range setting. Lower range settings will provide higher accuracy of measurement. The auxiliary port is configured as an analog voltage sensor **only** when no sensors have been enabled in the "Sensors" configuration tab.

3 or 4 Terminal Mode: This setting allows you to configure the instrument into a remote voltage sensing configuration for the RE+, RE- sense lines for use in 3 and 4 terminal applications.

Monitor Tab

The Monitor Tab is used to examine general operation and statuses of the selected instrument. A monitor window is added to this display by selecting a device in the device list and clicking on the “Add Monitor” button at the bottom of the list. One or more devices can be monitored simultaneously through this display window. The window has 3 main parts, status information is on the left, a running display of active levels is shown on the right and sensor information is in the center.

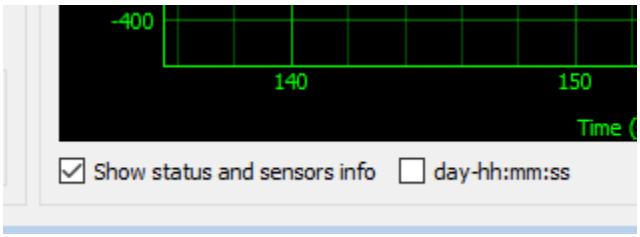


The status information box includes items such as which control the instrument is currently operating in, how long the test has been running, how long the active control has been running, and what the voltage, current and charge levels are. There is also a *Processing Utilization Meter*: The design of our instrument and software allows for a very wide range of configuration possibilities. As a result, it is not possible to determine at the PC software level whether your test may exceed the processing limitations of the instrument. The processing utilization meter is provided here to assist you in determining if your test design is reaching or exceeding the processing limits of the instrument and needs to be scaled back.

The Sensors box contains information on internal temperature levels inside the instrument, present levels of any temperature sensors that have are attached to the instrument and activated, and present levels of pressure sensors that are attached to the instrument and activated.

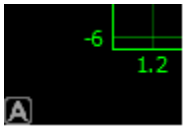
The Active Measurements window shows a running display of the Voltage and Current levels for the battery that is attached to the instrument. The plots show a record of up to 1000 data points. Older data points will automatically be purged from the plot. The Monitoring interval in the Design->Config Tab adjusts the update rate for the monitor window for each instrument.

The status information box and sensors box can be hidden/shown by unchecking /checking the “Show status and sensors info” box under the active measurements window.



The graph axis time supports two format: “seconds” and “days,hours:minutes:second”. The default format is in “seconds”. To switch to the “days,hours:minutes:second” format, check “days-hh:mm:ss”.

The graph axis can be adjusted using the mouse buttons and scroll wheel. See section on Mouse Shortcuts for details. Right click on a graph to bring up a menu of selections to change characteristics of the plot or to save the current plot displayed out to files of various formats, graphical or text. A graph that has been manually adjusted, can be reset to the default auto-scaling, auto-update mode by clicking on the “A” in the lower left hand corner of the graph.



Graphs

Impedance, Nyquist *(On supported models)*
These graphs are associated with AC impedance testing. They are used to display testing data for AC impedance sweeps performed during constant charge or discharge control nodes and AC voltammetry tests during Ramp based control nodes.

When an instrument is actively performing one of the AC test noted above and the instrument is actively selected in the instrument selection list, the results of the test will be plotted in real time to both Impedance and Nyquist graphs. The data from these tests will also be saved into Venable data files with the extension of “.vacz” regardless of which device is selected for monitoring. The default names assigned to the files during AC test contains the following: Chassis Number, Channel Number, Date and Time (Chas#_Chan#_Year_Month_Day_Hour_Minutes_Seconds). These files can be recalled into the graphs using the “Open ACZ file” button on the left bottom portion of the window. All displayed plots are listed in the listing box on the left side of the graph window and can be selected by clicking on the plot name in the list. You can also hide a particular plot by unchecking the check box for the desired plot.

The name of each plot can be modified by the user. In order to change the default name, double click on the file name in the listing box to highlight the text for

modification. Then enter a new name. This will also change the name of the file that the data set is stored to within the saved folder. It is recommended that you leave the file extension ".vacz" intact in your new name in order to identify this data as AC impedance data. Each plot label can be moved using the mouse to drag the label to a new position.

Export to ZView  *:* This button will export the selected data into a file compatible with the ZView software developed by Scribner Associates, Inc.

Tafel

This graph is associated with cyclic voltammetry testing. In order to use this graph, you will need to define a log that will capture voltage and current information at regular intervals. The data defined by the log will be stored into a file on the computer. The result data file can be loaded into the graph using the "Open Tafel Log file" button.

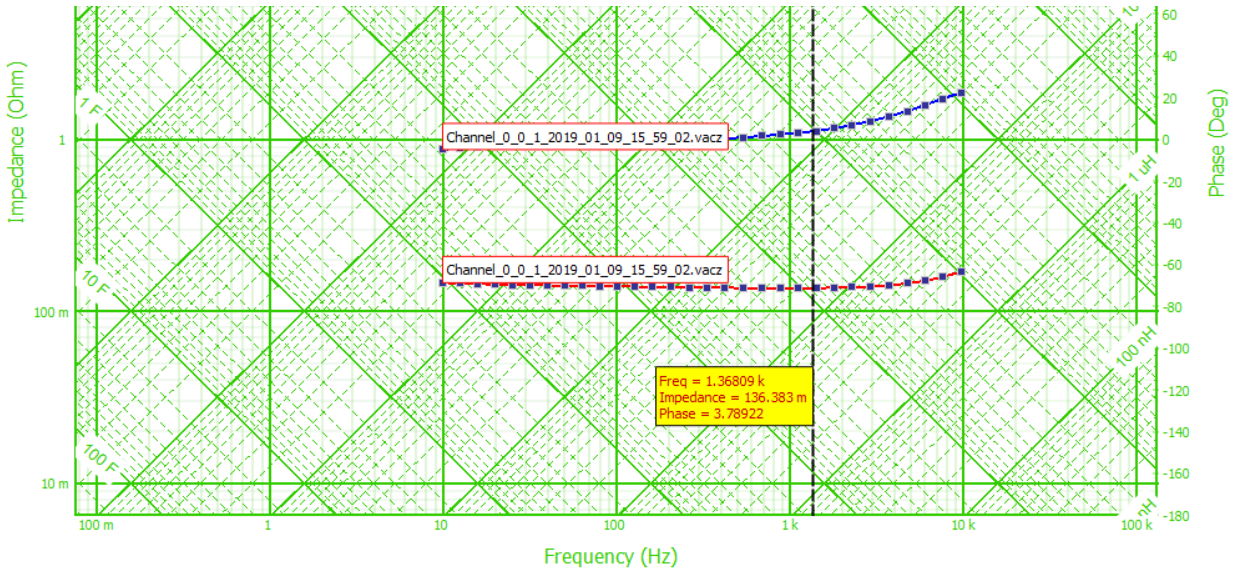
The default names assigned to the files during AC test contains the following: Chassis Number, Channel Number, Node ID, Date and Time (Chas#_Chan#_NodeId_Year_Month_Day_Hour_Minutes_Seconds).

The name of each plot can be modified by the user. In order to change the default name, double click on the file name in the listing box to highlight the text for modification. Then enter a new name. This will also change the name of the file that the data set is stored to within the saved folder. It is recommended that you leave the file extension ".vlog" intact in your new name in order to identify this data as logging data.

Add Slide Bar

This function is accessed through the "Add Slidebar" icon located on the top left of the graph. The slide bar is useful for tracking and displaying data values at a specific frequency, represented by a vertical line and a description box. The Add Slide Bar command can only be run when a single data set is selected. More than one slide bar may be opened at one time, either from the same or different data sets.

The bar can be dragged using the mouse pointer, and the box will contain the data at the selected frequency. The box can be dragged up or down along the slide bar using the mouse. To delete a slide bar, right click on the bar. This will show a delete button. Click on delete button to remove the slide bar.

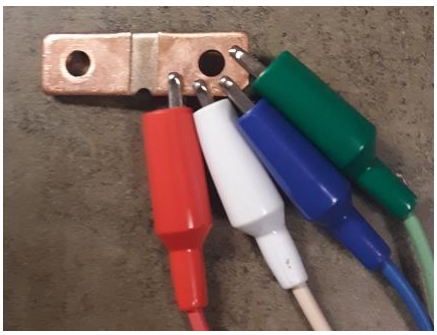


Method for Calibrating Impedance
Method for Extending Impedance Accuracy

The EstiView software includes a Math package to allow for correction of parasitics in EIS measurements introduced by the instrument itself and the attaching cables. It is also possible to achieve accuracy extension beyond the base Accuracy Contour Plot of approximately 2 decades. Access to these math tools can be done through either the Menu, Tools->Math, or through the Open Math Tool button at the bottom of the file list. The following procedure outlines the steps necessary to perform this correction on your measured impedance plots. This procedure will utilize two 100 microOhm current sense resistors to obtain open and closed loop plots for use in the math calculations. Venable recommends Vishay part: WSBS5216L1000JK for the best conductive quality material and insure the highest quality results.

<u>Measure Closed Loop Impedance:</u>

Connect all test leads across one side of the sense resistor as shown. Create a galvanostatic ACZ impedance sweep test with the current range set to 5 Amps and the Voltage range set to 1.8 V. The AC amplitude will need to be set to 3 Amps. Set desired Start and Stop frequencies. Run the test to plot and record the Closed Loop impedance.



Measure Open Loop Impedance:

Connect the Counter electrode and RE+ test leads across one side of one sense resistor. Then connect the Working Electrode and RE- across one side of the second sense resistor. This is demonstrated in the figures shown. NOTE: Do not allow the 2 sense resistors to come into contact with each other.
Create a potentiostatic ACZ impedance sweep test with the Voltage range set to 18V. Setting the current range will depend on the frequency range you are sweeping to insure that proper range is set to account active current amplitudes. For frequencies below 1 kHz, set the range to 5 microAmps. For frequencies above 1 kHz, set the range to 5 milliAmps. The AC amplitude will need to be set to 10 Volts. Set desired Start and Stop frequencies. Run the test to plot and record the Open Loop impedance.

Measure Device Impedance:
Generate an ACZ impedance sweep test appropriate to the device under test and perform the sweep to obtain the uncorrected measurement plot of the device. Choice of correction will now depend on where your device plot exists relative to the Open and Closed Loop plots. A low impedance device plot is too far below the open circuit impedance to require compensation for open circuit parasitics. Similarly, a high impedance plot is too far above the closed circuit impedance to require compensation for closed circuit parasitics.

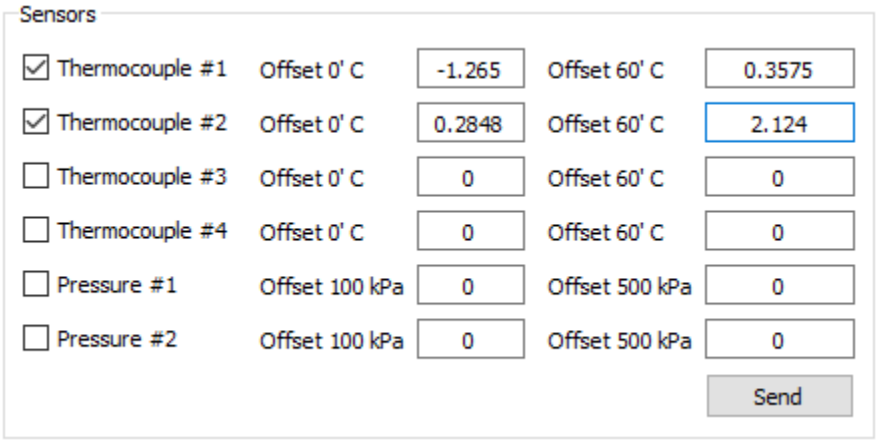
Low Device Impedance:
For this case, we correct for short circuit parasitics by subtracting the short circuit impedance from the test data. Open the Math Tool. For the Left Operand, select the bullet Data Set and select the Device data set. For the Right Operand, select bullet Data Set and select the closed loop plot data set. Set Actions selection to Subtract and click on the Plot button. The resulting graph will be the device impedance corrected for parasitics.

.

High Device Impedance:
The same concept as Low Device Impedance can be used for measuring high impedances and correcting for open circuit parasitics, except it is a little trickier because the open circuit parasitics are in parallel with the test data instead of in series. Essentially, what you want to do is subtract the *admittance* of the open circuit parasitics from the *admittance* of the test data (which includes the impedance of the device under test in parallel with the parasitic open circuit impedance). We have a feature in the Math Options called Subtract in Parallel. This will put the negative of the parasitic impedance in parallel with the test data, effectively canceling out the open circuit parasitic impedance and yielding only the impedance of the device under test.

Open the Math Tool. For the Left Operand, select the bullet Data Set and select the Device data set. For the Right Operand, select bullet Data Set and select the open loop plot data set. Set Actions selection to Subtract in Parallel and click on the Plot button. The resulting graph will be the device impedance corrected for parasitics.

Sensors Tab

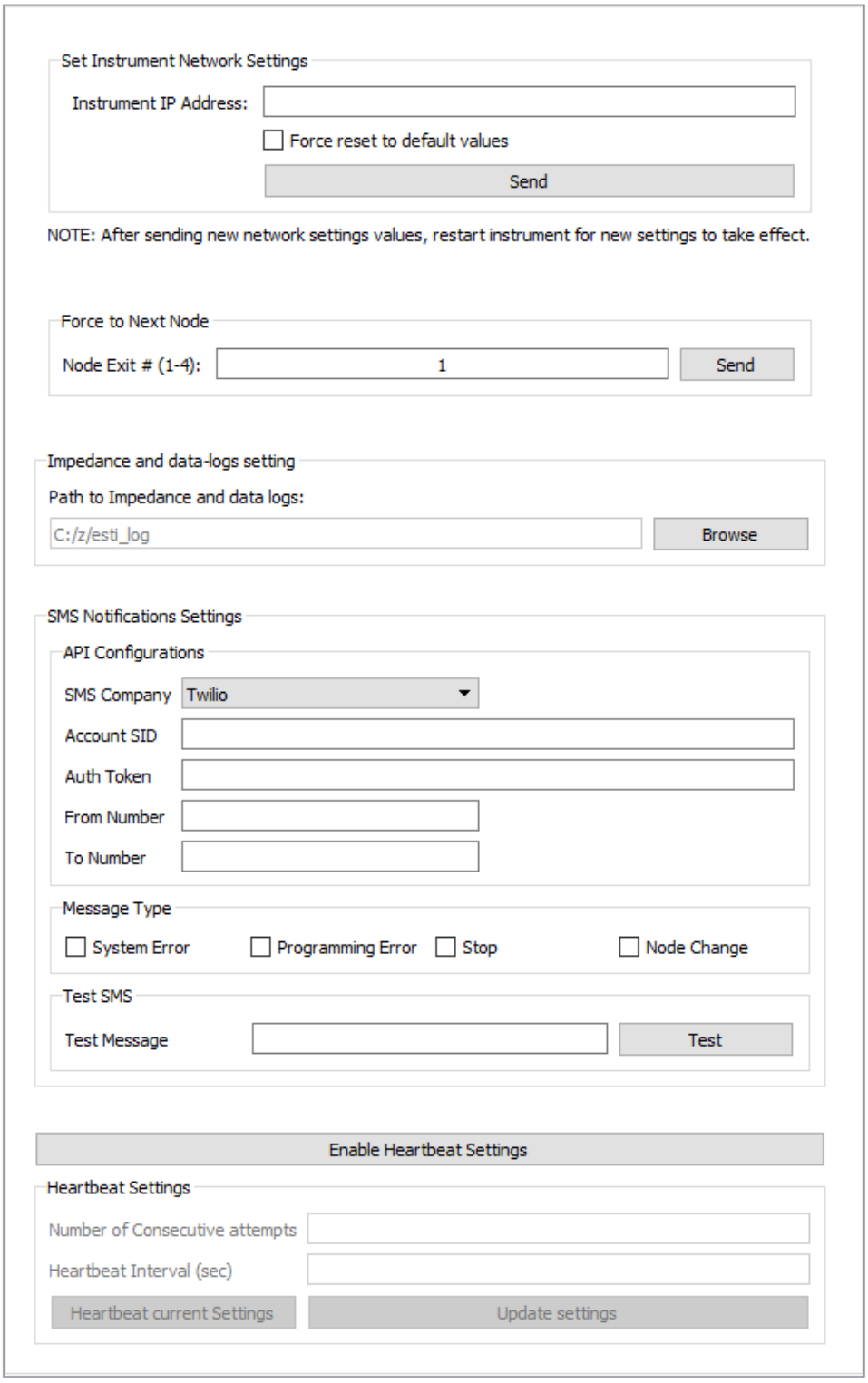


This window provides the ability to activate various optional sensors that have been attached to the instrument and provide a user calibration of the sensors. Click on the check box on the left side to activate a particular sensor. A linear calibration of the sensor is provided in this window as well. For thermal sensors, you will need to determine the error of the sensor against a known accurate standard at two temperature levels: 0 degrees Celsius and 60 degrees Celsius. The amount of error relative to the standard, positive or negative, is entered into the two respective fields associated with the sensor. This will provide you a good calibration of the sensor in a range of -40 degrees Celsius to 100 degrees Celsius. Once sensors have been selected and calibration information has been entered, you can transmit this information to the instrument by clicking on the “Send” button. Once this is done, the instrument will be configured with this sensor setup until the next power cycle is performed on the instrument.

NOTE: Calibration for higher temperatures is possible using interpolation methods of the 60 degree offset against a measurement of a standard at the higher temperatures.

Utilities Tab

This window contains various instrument configuration utilities that are generally unrelated to the design and testing operations of the instrument.



Set Instrument Network Settings:
This can be used to change the Ethernet IP address of the instrument. New ESTi instruments are delivered with a default IP address 192.168.214.214. If you have multiple instruments, this utility can be used to set each instrument to a unique IP address. After entering your updated information in the box, click on the “Send” button to update your instrument with the new IP address information. The new settings will take effect after cycling the power on the instrument.

Note: You must be able to connect to the instrument prior to any attempt to change and send the new IP address information.

There is also a check box that will configure the instrument back to the default values above. If this box is checked, the information in the address box above it will be ignored.

Force to Next Node:
This provides you the ability to force operation of the selected instrument to the next node in your test setup. Enter the Exit path number (1-4) into the box, corresponding to the path you wish to take out of the active node. Then push “Send”. The selected instrument will immediately transition to the new node for operation.

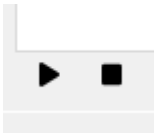
Impedance and data-logs settings:
This allow you to choose the folder path to the location on the PC where your data logging files and Impedance data output files are stored to.

SMS Notifications:
This section allows you to set up texting notifications based on events occurring in the test instrument. Supported providers are listed in the SMS Company pulldown. You will need to set up an account with a supported provider in order to enable this feature. Once an account is setup, you will need to select the provider and then enter your username, password and subscription-provided phone number into the fields. Enter the receiving phone number in the “To Number” field for text messages to be sent to. The texting setup can be verified by entering a test message and then clicking on the “Test” button. You should receive the test message on your cell phone or other texting device.

A selection box is provided to choose the events that will trigger a text message to be sent to you. The message will include identification of the instrument for which the event originated from.

Heartbeat Settings:
This section allows you to configure the heartbeat interval and the number of consecutive beats/attempts. The heartbeat is a common technique for monitoring the connection between the instrument and the computer and detecting connection breaks. The default heartbeat interval for new ESTi instruments is 5 seconds, and the number of consecutive attempts is 4. By clicking on the "Heartbeat Current Settings," the software will display the current heartbeat settings in the boxes.

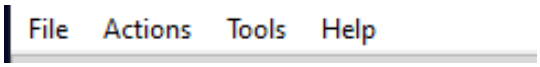
Shortcut icons



There are short icons for Start Test and Stop test on the bottom left of the software window. To use these shortcuts, you must have an active connection to an instrument, have selected the instrument from the selection list and have already loaded a valid test scenario into the selected instrument. Click on the Start Test icon to initiate the test. The Stop Test icon can be used to abort and end the testing operation completely for the selected instrument.

***NOTE:** Do not "Start" a test unless a valid test scenario has been loaded into the instrument. Otherwise damage to the instrument or device under test could occur.*

Main Menu



File

New Test

This selection will erase any test created in the node layout and connection editor. It will also delete any formulas that have been created and any Logs that have been defined. It will also reset parameters in the Config tab back to their default values.

Open Test

This selection will open a file selection dialog box. Navigate to the file containing a saved test configuration. Double click on the file to select it and load the test into the node layout and connection editor. Formulas, Logs and Config information are also saved in the file and will be restored in their respective tabs.

Save Test

This selection will store the test configuration created with the node layout and connection editor, the conditional assignments associated with condition links, all Formula definitions and all Logs defined. This selection will also save parameters defined in the Config tab.

Print

Prints the active graph, including titles, curves and slide bars to the selected local or network printer.

Print Preview

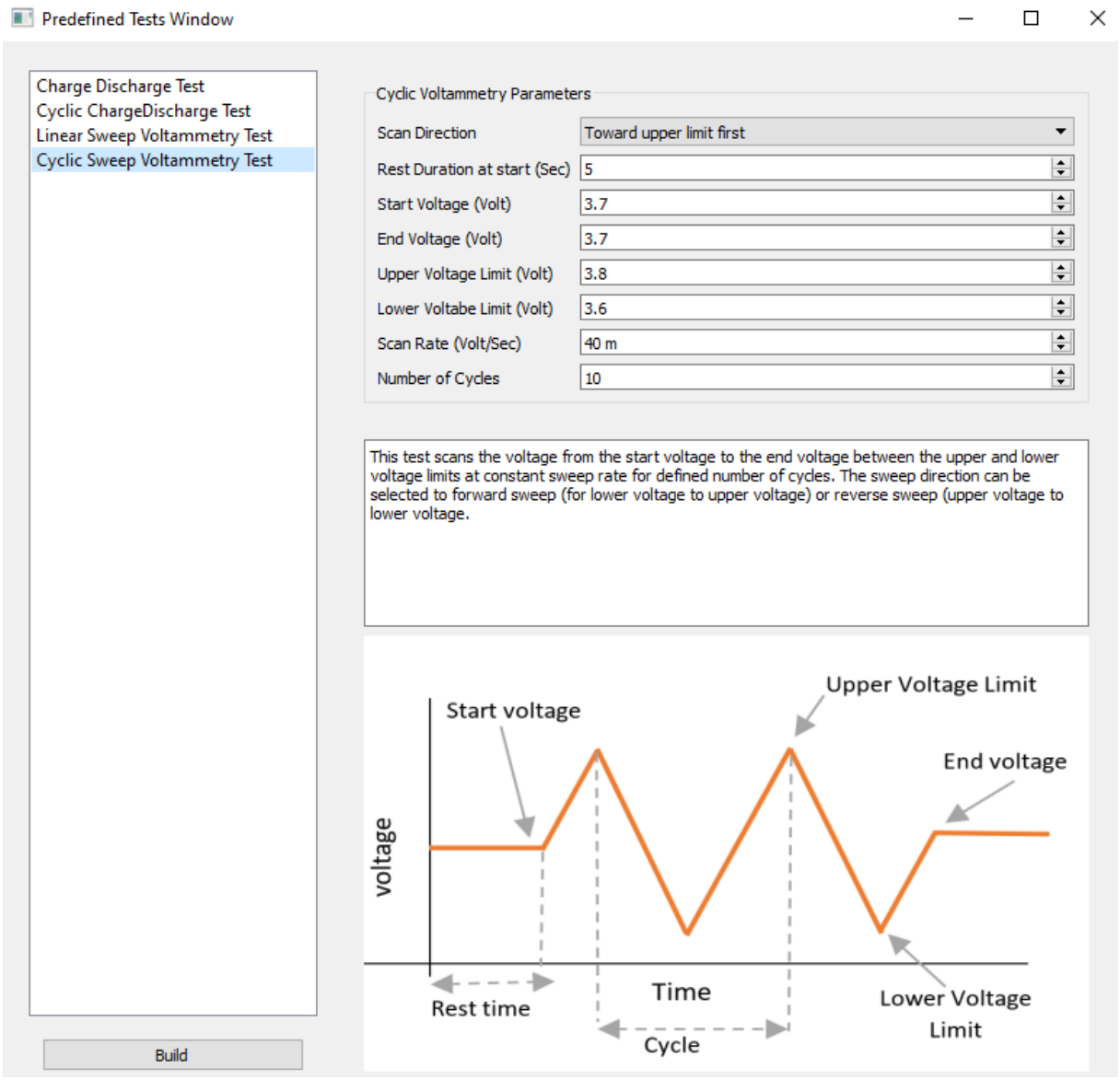
Displays what the printed page will look like. This is recommended before you print.

Actions

Predefined Tests

This will display the predefined tests window. This window allows you to build the most common electrochemistry tests easily and quickly.

The panel on the left shows the available tests, while the panel on the right shows the test parameters settings, the test description, and the applied waveform to the cell.
Once you select the test and configure its parameters, click on the build button. The software will build the selected test.



Connect

This will display the connection dialog. Enter the IP address of the instrument you want to connect to and click on the connect button. (Instrument Port value should remain set to the default value). When the software has successfully connected to the instrument, the Status display box will show a valid connection.

Load Test

This will download a defined test scenario into the selected instrument. The download will include all node and condition definitions, all defined formulas and all defined logs, including their respective condition tests. It will also download the parameter data from the Config tab. Sensor activations and settings are not included in this function and must be performed separately. New tests can be loaded while the instrument is running a test. However, the transition to the new test will not occur until the active node exits. Care must be taken that the new test contains a node number corresponding to exit node of the active node in the old test.
The software provides a function to validate the test. When you click "load Test", the function will check all the nodes connections and exit conditions. If the test is valid the test will be loaded to the instrument, If not the software will show either an error message or waring message. If the software shows an error message, the test will not be loaded to the instrument.

Start Test

This will initiate testing activity for the selected instrument. The software must have an active connection established to the selected instrument and have a valid test scenario downloaded into the instrument.

NOTE: *Do not "Start" a test unless a valid test scenario has been loaded into the instrument. Otherwise damage to the instrument or device under test could occur.*

Stop Test

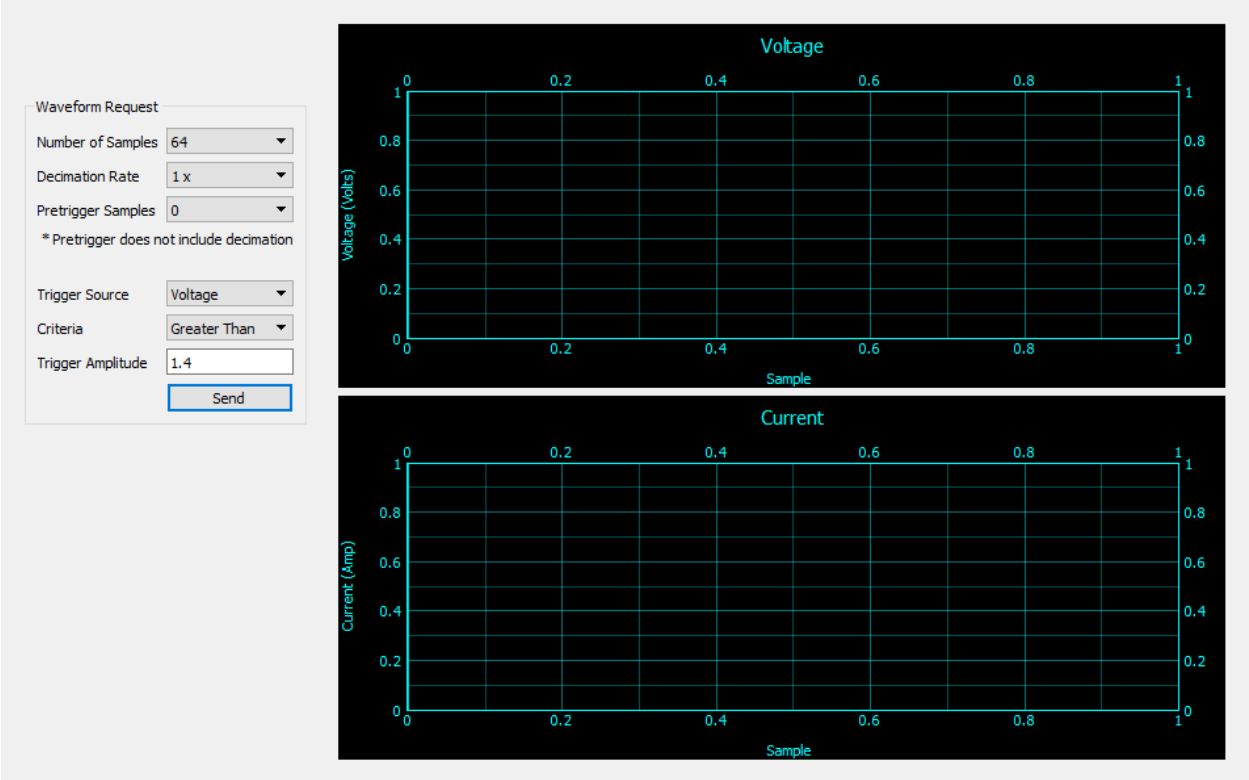
This can be used to abort and end the testing operation completely for the selected instrument.

Capture Waveform

This item essentially provides an oscilloscope capability for the instrument and software. Selecting this menu item will open a separate window containing two graphing windows to show plots of voltage and current levels. On the left side of the plots are controls to configure the instrument for a data capture. This function captures detailed data at the sampling rate set in the instrument. For constant charge/discharge operations, the sampling rate is selectable up to 50 kHz. Ramp and Pulse functions can be set to higher sampling rates of up to 208 kHz.

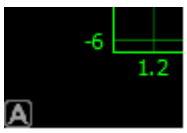
The capture will be performed on the selected (highlighted) device in the connection list.

To perform a data capture and display, select the total number of samples you wish to capture, a decimation rate that defines the rate at which to skip samples and then number of samples to collect prior to the trigger point. Then select whether to trigger on a voltage or a current level, the trigger direction, positive or negative, and the amplitude of the trigger point.



Click on the “Send” button to engage the request with the chosen trigger point. When the instrument reaches the trigger point, the data will be sent back to the software and immediately displayed in the plot windows. Once the trigger has occurred and the plots are displayed, the trigger can be immediately set again by clicking on the “Send” button again.

The graph axis can be adjusted using the mouse buttons and scroll wheel. See section on Mouse Shortcuts for details. Right click on a graph to bring up a menu of selections to change characteristics of the plot or to save the current plot displayed out to files of various formats, graphical or text. A graph that has been manually adjusted, can be reset to the default auto-scaling, auto-update mode by clicking on the “A” in the lower left hand corner of the graph.



Tools

Math

This will open the Math tool window. This tool is used for correction of parasitics in EIS measurements introduced by the instrument itself and the attaching cables.

Update firmware…

This will open the Venable ESTi Application Firmware Update tool. This is an executable file and is used to update the ESTi instrument firmware.

Help

Venable Help

This will open an interactive help window detailing operation of the software.

Venable Manual

Opens the Venable System software manual, which is located in the program directory with the Venable System program executable. This file is in Adobe portable document format (pdf) and is compatible with Adobe Acrobat Reader. An installation program for Adobe Acrobat Reader can be downloaded at www.adobe.com.

License…

This will open the software licensing tool in a separate window. The licensing tool shows the currently active license being used and provides a button, "Change License", to restart the Venable License Wizard (Administration rights required). A separate manual, titled “Venable Software License Manual” is provided and details use of the licensing software.

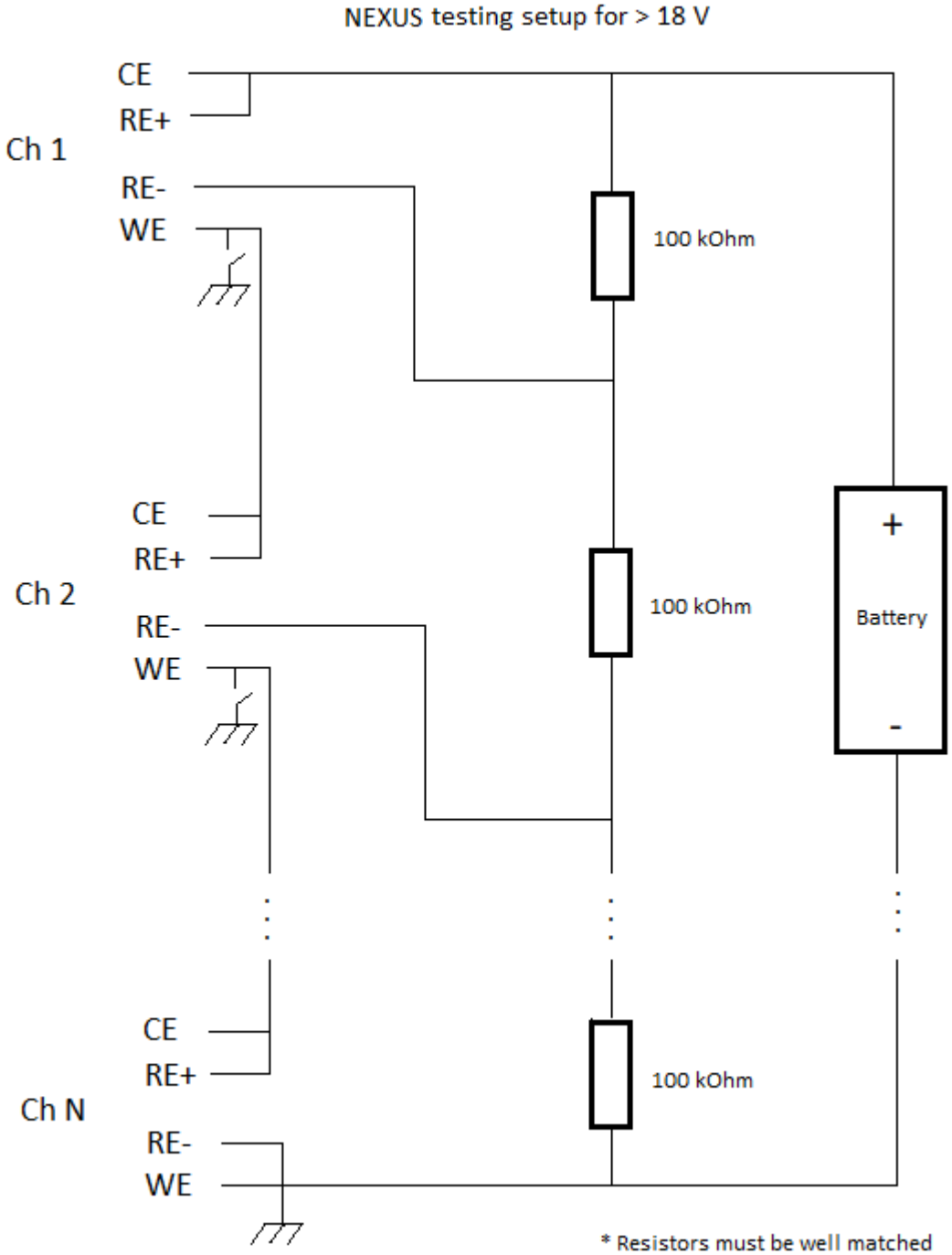
About Venable…

This opens a separate window, with the Venable Industries logo, that includes the software version number, internet address and general copyright information.

Nexus Synchronization

Serial Connection of channels (Stack Up)

The following diagram show method to connect channels in a serial configuration to boost voltage. It is not recommended to exceed the maximum number of channels listed in each testing method to insure the stability of the stack before and during testing.



EIS Testing

Nexus channel can be connected in a serial configuration with a maximum of 8 channels to boost voltage capacity. This provides a maximum battery limit of 140VDC. Operational stability of the stack improves with fewer channels, so it is recommended to use only as many channels in a serial configuration as needed to meet your voltage requirement.

When connecting to channels with the EstiView software, all channels in the stack must have chassis letter selected and the synchronization enabled (Synced: checkmarked)

The following constraints in the test node configuration must be met as a minimum:

Potentiostatic:

Ch 1 (master): Set to Voltage Positive. Disable PID. Set DC Amplitude = Voc /N

Ch 2 thru N (slaves): Set to Voltage Positive. Enable PID.

Set DC Amplitude = Voc/N

Galvanostatic:

Ch 1 (master): Set to Current Charge. Set DC Amplitude = 0.

Ch 2 thru N (slaves): Set to Voltage Positive. Enable PID.

Set DC Amplitude = Voc/N

When building the node diagram, the first node out of the Begin node must be as follows:

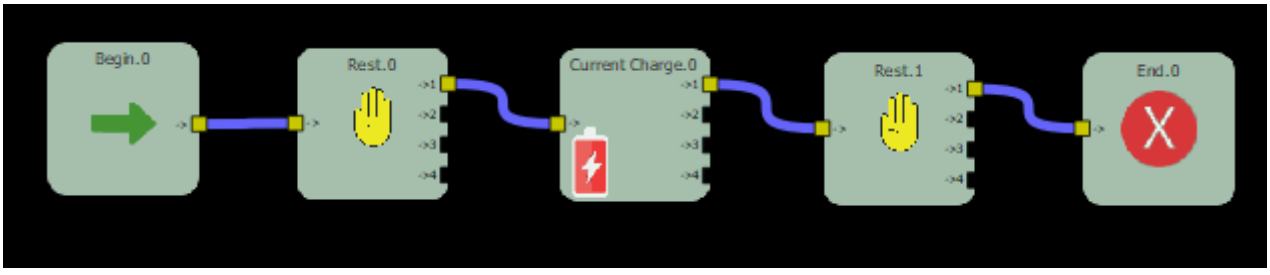
Ch 1 (master): Rest Node

Ch 2 thru N (slaves): Voltage Positive Node

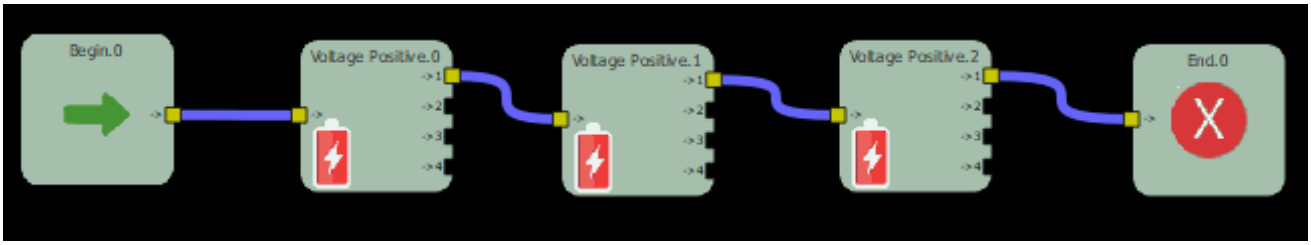
This is to insure that the stack can stabilize before entry into active operations by the master controller. Exit from the master Rest node should be set to minimum 2 second delay. A rest node on master should also exist just prior to exit of the test to allow slave channels to shut down gracefully

Care must be taken to insure that master total path node IDs match all slave channels total path node IDs. Node IDs are not shown in the icon. You must double click on the icon itself and the node IDs will be shown at the top of the Node Editor. Below is a basic example of master channel and slave channel node configurations. Master channel Current charge contains a galvanostatic ACZ sweep operation.

Master Channel:



Slave Channels:



Following loading of the test, one or more of the channels might exceed voltage limits and transition into a “Stop with Error”, outlined in red. In order to eliminate this issue, set the Shutoff Voltages in the Config tab to -20V for min and +20V for max. This will set the limits above the maximum measuring limit of the channel and prevent transition into the error state.

Stack-up EIS provides accurate measurement for frequencies at or below 500 Hz. Measurements above 500 Hz are adversely affected by the resistor divider network. Injection amplitude should typically remain below 1 amp. Injections higher than this could affect resistor network causing resistor accuracy drift. It is generally recommended to use the lowest injection ampltude possible that still provides a clean result with little noise in the plot.

Results:
Phase plot does not require any adjustment.
Impedance plot must be multiplied by the number of channels in the serial stack to obtain the actual impedance of the battery. This can be done with the Math Tools menu.

Keyboard and Mouse Shortcuts

Like any Windows program, you can hold down the Alt key and hit any underlined letter to activate that pull-down menu. Many of the standard Windows Edit Menu control key functions also work, for example:

Ctrl+C	Copy Data Set
Ctrl+V	Paste Data Set
Ctrl+X	Cut Data Set
Ctrl+Z	Undo Last Edit

The software has keyboard shortcuts associated with it and selecting the parameter with the mouse.

Up or Down arrow key	Increments parameters in various parameter editing windows according to preset multiplier or integer step sizes.
Page Up or Page Down key	Increments parameters in various parameter editing windows according to preset integer step sizes.

Besides the keyboard shortcuts, there are mouse short cuts.

1) Left click on any point of any data set to select it.
2) Left click and drag any window or menu to move it or dock it.
3) Left click and drag the side or corner of a window or menu to resize it.
4) Use scroll wheel OR right click and drag the graph to the left of the Impedance or right of the Phase axes to change the gain or phase scaling respectively.
5) Left click and drag the graph to the left of the Impedance or right of the Phase axes on the Impedance or Phase axes to change the gain or phase range displayed.
6) Use scroll wheel OR right click and drag the graph anywhere between Gain and Phase axis to change frequency scaling.
7) Left click and drag the graph anywhere between Impedance and Phase axis to change range of frequency values displayed.
8) Use scroll wheel to change modifiable mouse selected parameters according to preset step sizes.
9) A graph that has been manually adjusted, can be reset to the default auto-scaling, auto-update mode by clicking on the “A” in the lower left hand corner of the graph

Appendix A

Direct Computation

The following illustrates an example of implementing a quadratic equation:

Given: $y = (A * x^2) + (B * x) + C$

For this demonstration, we assume x is Voltage,
And the result 'y' is stored in user defined variable "QuadraticResult".

Create Variables: IntermediateCalculation = Voltage * B + C
QuadraticResult = Voltage * Voltage * A + IntermediateCalculation

To insure that the formula is calculated correctly, remember to implement the order of processing correctly. Intermediate values must be defined first to be used in subsequent calculations.

Conditional Assignment

Let's assume we want to do the following comparison and assignment as illustrated in pseudo-code

```
If Voltage > Vmax then
    Vmax = Voltage
```

Voltage is selected for Variable A. Vmax is the new variable. The above comparison can then be rewritten as follows for implementation into a conditional assignment:

```
Vmax = Voltage > Vmax
```

For a more complex expression such as,

```
If  ( ( Voltage > Vmax ) && ( Current  >=  2.5 ) )  then
    Vmax = Voltage
```

We can rewrite the function as:

```
Vmax = Voltage > Vmax AND Current >= 2.5
```

Here, the value of Voltage is put into user-defined variable Vmax when the entire string evaluates to TRUE.

Timed Recursive Log Example

This demonstrates how to define the formulas and logging conditions to perform a log of data at 5 second intervals.

Create 2 formulas:

Direct Computation (Integer)

DelayedTime = Time Into Test – 5.0

Conditional Assignment (Integer)

TriggerTime = (Time Into Test > 0.0) And (DelayedTime >= TriggerTime)

Create Log Condition for the relevant Log:

TriggerTime = Time Into Test

Appendix B

Calibration Specifications for 21xx, 23xx, 27xx and 29xx ESTi / Nexus instruments

THERMAL:
Adjustment performed at 1ºC, 27ºC and 55ºC ambient
Calibration (verification) performed at 14ºC and 38ºC ambient

OUTPUT DC:

Potential:	18V (1 range only):	+/- 18mV
Current:	All Ranges:	0.2% of FSR

OUTPUT AC: (on equipped models)

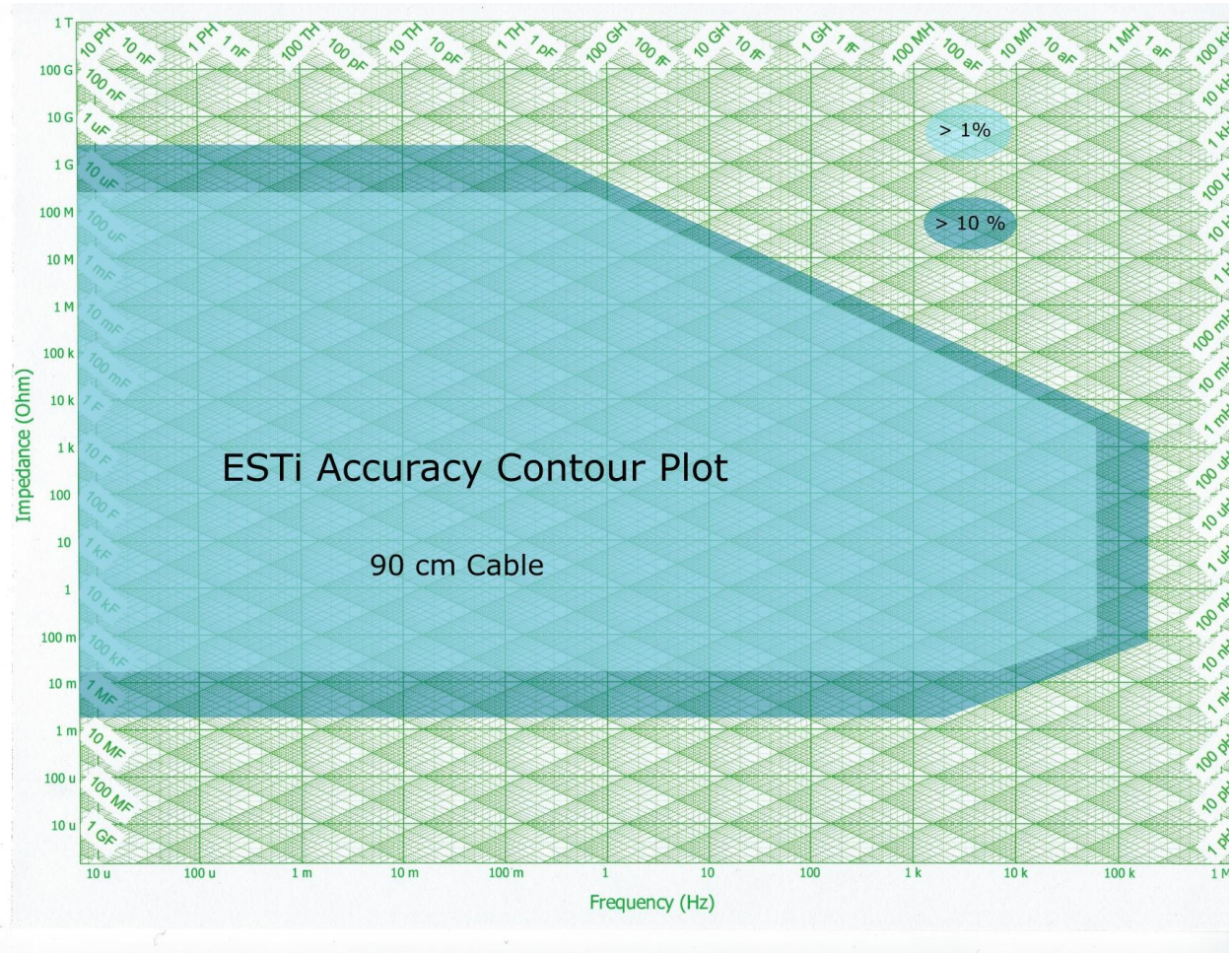
Potential:	18V (1 range only):	0.5% of FSR
Current:	5A, 150mA and 5mA Range:	0.2% of FSR
	150µA and 5µA Range:	0.5% of FSR

MEASUREMENT, DC and AC:

Potential:	18VRange:	0.2% of FSR
	1.8V Range:	0.26% of FSR
	180mV Range:	0.4% of FSR
	18mV Range:	1.0% of FSR
Current:	All Ranges:	0.2% of FSR
Auxiliary:	20V and 2V Ranges	0.2% of FSR

AC Measurement Note:
Measurements are calibrated in real time at the sampling layer (each individual sample). As a result, only calibration to DC standards is required and provides more accurate results for AC measurements.

AC Impedance Accuracy Contour Plot:



Appendix C

Hardware Description of ESTi instruments

2161, 2165, 2185, 2361, 2365, 2385 Front Panel

Description

1. Status OK: This green LED Illuminates when the power is turned on and the device has completed boot up operation.
2. Enabled: This yellow LED Illuminates the output is active.
3. Auxiliary Jack: Provides for additional voltage measurement connection, or for supported digital sensors.
4. Main Jack: Provides signal connection for the Working and Counter Electrode contacts and the electrometer reference contacts.
5. Power Switch: This switch turns the instrument on and off.
6. Button: Not implemented.

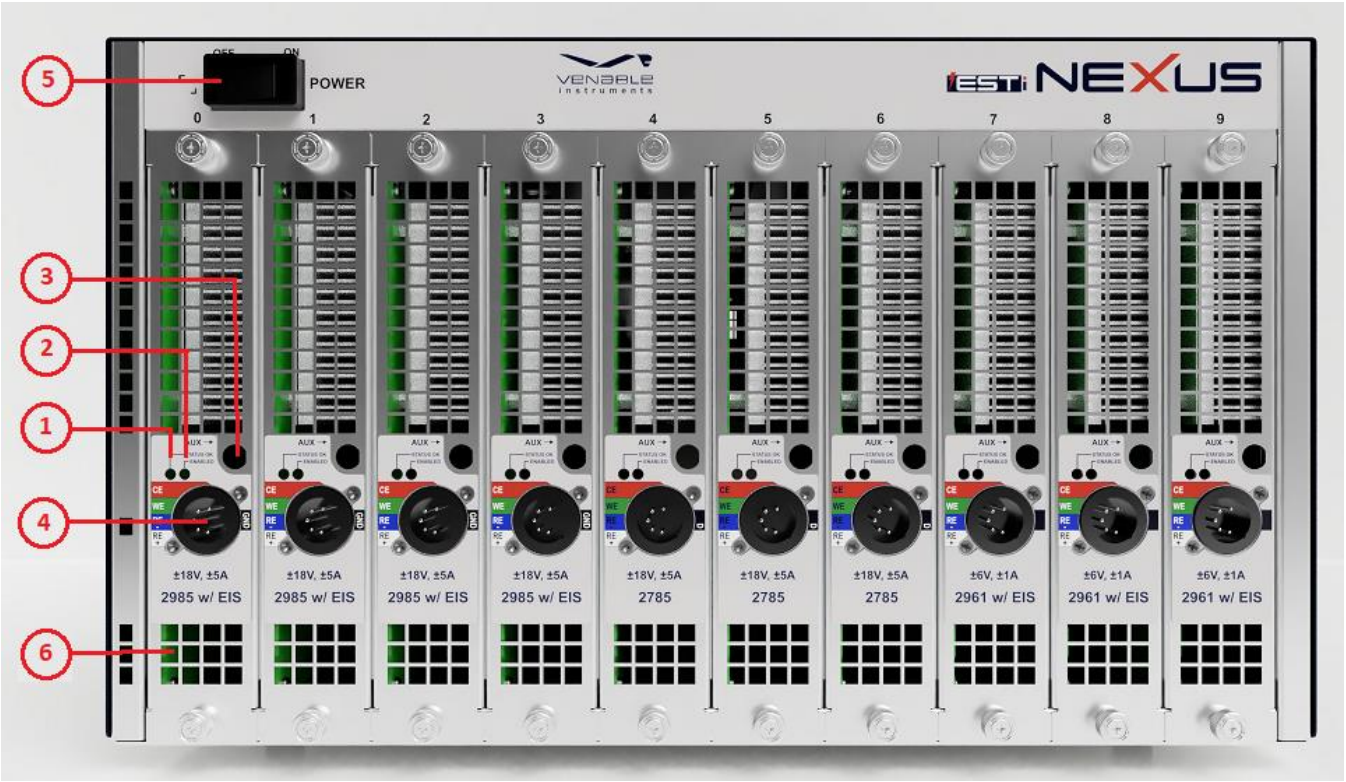
2161, 2165, 2185, 2361, 2365, 2385 Rear Panel

Description

1. Power Inlet: Supplies the required 90 to 264Vac, 48 to 62Hz, and 150VA power to the instrument.
2. Fuse: Fuse holder for 3A/250V slow-blow fuse.
3. Ethernet Port: Provides Ethernet communication with the Venable ESTiView software.
4. Cooling Fans NOTE: Do not block airflow.
5. Ground Plug When installed, this connects the instrument to earth ground. The plug can be removed to connect the instrument to a floating reference.

Hardware Description of Nexus instruments

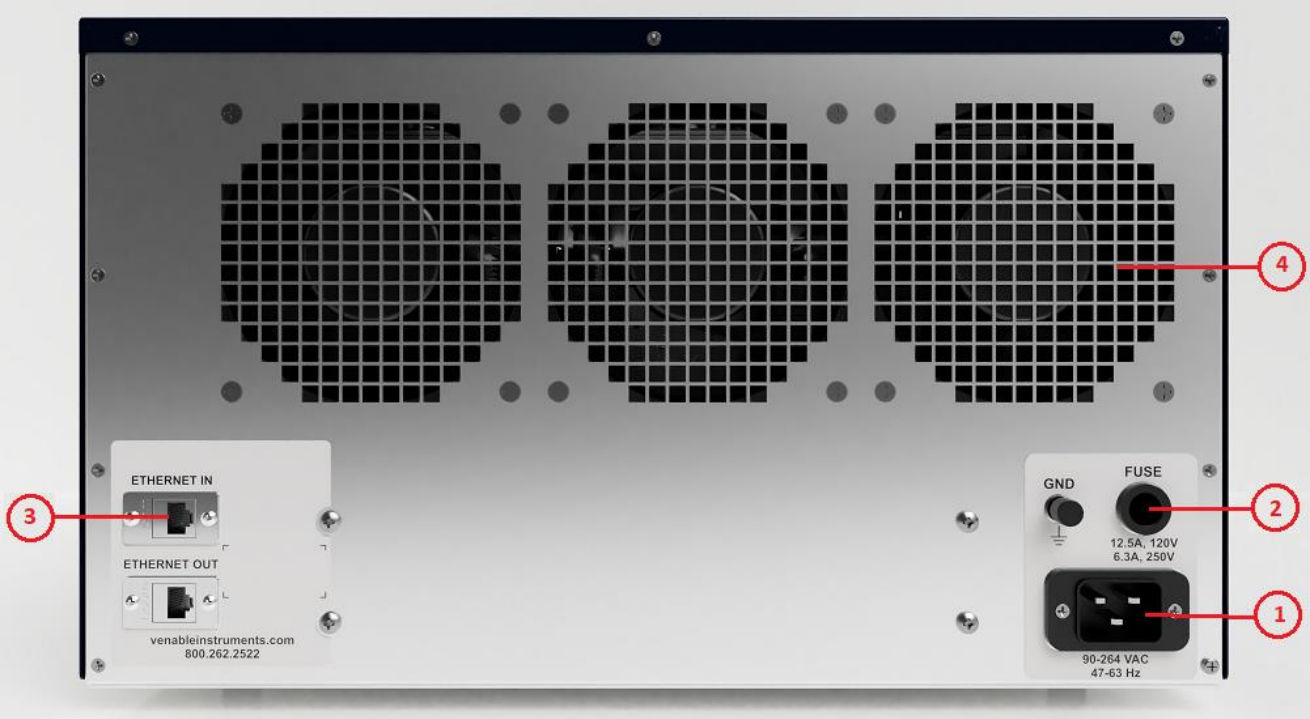
Front Panel



Description

1. Status OK: This green LED Illuminates when the power is turned on and the device has completed boot up operation.
2. Enabled: This yellow LED Illuminates the output is active.
3. Auxiliary Jack: Provides for additional voltage measurement connection, or for supported digital sensors.
4. Main Jack: Provides signal connection for the Working and Counter Electrode contacts and the electrometer reference contacts.
5. Power Switch: This switch turns the instrument on and off.
6. Button: Not implemented.

Nexus Rear Panel



Description

1. Power Inlet: Supplies the required 90 to 264Vac, 48 to 62Hz, and 1500VA power to the instrument.
2. Fuse: Fuse holder for 12.5A/120V or 6.3A/250V slow-blow fuse.
3. Ethernet Port: Provides Ethernet communication with the Venable ESTiView software
4. Cooling Fans NOTE: Do not block airflow.