Racal Instruments™

3172

VXIbus Arbitrary Waveform Generator and Dual Pulse Generator

User Manual

Includes various 3172 configurations

Publication No. 980949 Rev. L

Astronics Test Systems Inc.
4 Goodyear, Irvine, CA 92618
Tel: (800) 722-2528, (949) 859-8999; Fax: (949) 859-7139

atsinfo@astronics.com atssales@astronics.com
atshelpdesk@astronics.com http://www.astronicstestsystems.com

THANK YOU FOR PURCHASING THIS ASTRONICS TEST SYSTEMS PRODUCT

For this product, or any other Astronics Test Systems product that incorporates software drivers, you may access our web site to verify and/or download the latest driver versions. The web address for driver downloads is:

http://www.astronicstestsystems.com/support/downloads

If you have any questions about software driver downloads or our privacy policy, please contact us at:

atsinfo@astronics.com

WARRANTY STATEMENT

All Astronics Test Systems products are designed to exacting standards and manufactured in full compliance to our AS9100 Quality Management System processes.

This warranty does not apply to defects resulting from any modification(s) of any product or part without Astronics Test Systems express written consent, or misuse of any product or part. The warranty also does not apply to fuses, software, non-rechargeable batteries, damage from battery leakage, or problems arising from normal wear, such as mechanical relay life, or failure to follow instructions.

This warranty is in lieu of all other warranties, expressed or implied, including any implied warranty of merchantability or fitness for a particular use. The remedies provided herein are buyer's sole and exclusive remedies.

For the specific terms of your standard warranty, contact Customer Support. Please have the following information available to facilitate service.

1. Product serial number
2. Product model number
3. Your company and contact information

You may contact Customer Support by:

E-Mail:	atshelpdesk@astronics.com	
Telephone:	+1 800 722 3262	(USA)
Fax:	+1 949 859 7139	(USA)

RETURN OF PRODUCT

Authorization is required from Astronics Test Systems before you send us your product or sub-assembly for service or calibration. Call or contact Customer Support at 1-800-722-3262 or 1-949-859-8999 or via fax at 1-949-859-7139. We can also be reached at: atshelpdesk@astronics.com.

If the original packing material is unavailable, ship the product or sub-assembly in an ESD shielding bag and use appropriate packing materials to surround and protect the product.

PROPRIETARY NOTICE

TRADEMARKS AND SERVICE MARKS

DISCLAIMER

Buyer acknowledges and agrees that it is responsible for the operation of the goods purchased and should ensure that they are used properly and in accordance with this document and any other instructions provided by Seller. Astronics Test Systems products are not specifically designed, manufactured or intended to be used as parts, assemblies or components in planning, construction, maintenance or operation of a nuclear facility, or in life support or safety critical applications in which the failure of the Astronics Test Systems product could create a situation where personal injury or death could occur. Should Buyer purchase Astronics Test Systems product for such unintended application, Buyer shall indemnify and hold Astronics Test Systems, its officers, employees, subsidiaries, affiliates and distributors harmless against all claims arising out of a claim for personal injury or death associated with such unintended use.

FOR YOUR SAFETY

Before undertaking any troubleshooting, maintenance or exploratory procedure, read carefully the **WARNINGS** and **CAUTION** notices.

This equipment contains voltage hazardous to human life and safety, and is capable of inflicting personal injury.

If this instrument is to be powered from the AC line (mains) through an autotransformer, ensure the common connector is connected to the neutral (earth pole) of the power supply.

Before operating the unit, ensure the conductor (green wire) is connected to the ground (earth) conductor of the power outlet. Do not use a two-conductor extension cord or a three-prong/two-prong adapter. This will defeat the protective feature of the third conductor in the power cord.

Maintenance and calibration procedures sometimes call for operation of the unit with power applied and protective covers removed. Read the procedures and heed warnings to avoid “live” circuit points.

Before operating this instrument:

1. Ensure the proper fuse is in place for the power source to operate.
2. Ensure all other devices connected to or in proximity to this instrument are properly grounded or connected to the protective third-wire earth ground.

If the instrument:

- fails to operate satisfactorily
- shows visible damage
- has been stored under unfavorable conditions
- has sustained stress

Do not operate until performance is checked by qualified personnel.

Table of Contents

This page was intentionally left blank.

.

Table of Figures

This page was intentionally left blank.

List of Tables

DOCUMENT CHANGE HISTORY

Revision	Date	Description of Change
A	10/22/2010	Document Control Release
B	7/14/2011	Updated per EO 30562. P2 information and configurations added. Existing information updated as necessary.
C	9/13/2012	Updated per ECN00895 - In Chapter 7, removed requirement for calibration steps 1 through 11 during routine, periodic calibration.
D	11/19/2012	Updated per ECN01917 – In Appendix A, added offset accuracy specifications for 93 ohm and high-impedance modes.
E	10/16/2013	Updated per ECN03713 - Added instructions for installing local bus jumpers. Added reference to these instructions in each place where local bus features are mentioned. In Chapter 5, rewrote introduction to clarify the methods of synchronization.
F	3/6/2014	Updated per ECN04225- Added details and clarification to calibration and firmware update procedures. Added calibration parameters for W6-type waveform generator module. Corrected the descriptions of EXT SCLK to indicate that it is compatible with PECL and NECL amplitude levels. Specified the periodic calibration interval in Chapter 7.
G	5/7/2014	Updated per ECN04838. Revised calibration procedure in Chapter 7, PLL Adjustments for cleaner signals and improved accuracy. Revised Chapter 6, PLL Checks – Phase Offset for correct length of arbitrary waveform. Added A3 amplifier documentation, including description, commands, performance checks, and calibration procedure.
H	8/19/2014	Updated per ECN05013. Revised Chapter 7, Table 7-3 to correct the W6 programmed amplitudes and expected responses for several calibration steps.
J	9/7/2017	Updated per ECN08890. Adds differences between W2 and W6. Explains 3171 emulation improvements. CLKOUT termination change. Includes NEG mode voltage specification change.
K	12/18/17	Updated per ECN09678. Mentions W6A3 configuration and common configurations. Updates Figure 1-1 with current photos. Deprecates W2 but includes all reference information. Corrects various text errors. Updates Controlled Pulse Parameters in Appendix A and Tables 6-38, 6-44, 6-45, 6-52, 6-58, and 6-59
L	1/3/2018	Updated per ECN09797. Updated Neg. mode and Period Accuracy specifications on pages A-3 and A-14. Reverted test limits in Tables 6-38 and 6-52 back to 3% limits.

This page was intentionally left blank.

Chapter 1
Introduction

What's in this Manual

This manual contains information for operating and servicing the Racal Instruments™ 3172 series of VXIbus Arbitrary Waveform/Pulse Generators. Generally, what is applicable to the 3172 is the same for all other models, except where otherwise stated. Throughout this manual, we refer to all units as the 3172. When there are differences in the specific models, the differences are described in detail.

More specific information on the 3172 models is included in the section, "3172 Configuration (Different Model Numbers)" later in this chapter.

Legacy mode information can be found in Chapters 3 and 5.

The manual is divided into functional chapters which guide you through the various operations that are necessary to install and to prepare the instrument for its intended operation. The following lists the chapters that are included in this manual:

- Chapter 1 provides general description of the instrument and identifies key controls and features. It also describes briefly all functions and features that are available for the user.
- Chapter 2 describes hardware and software installation.
- Chapter 3 provides descriptions of all functions, features, run modes and operating modes. It also describes in detail how to operate the instrument.
- Chapter 4 demonstrates the capability of ArbConnection to control the 3172 and to create and download waveforms and control tables to the working memory.
- Chapter 5 lists all of the commands that control the instrument. It also has detailed descriptions of the limits and factory default values of the programmable parameters.
- Chapters 6 and 7 contain service information that allows you to do performance tests and to calibrate the product.
- Appendix A lists the 3172 product specifications.

What's in this Chapter

This chapter contains a general and functional description of the Racal Instruments 3172 VXIbus Arbitrary Waveform Generator. It also explains the front panel connectors, operational modes, and all available features. However, some options available for the 3172 may not be installed in your specific module. A complete listing of the available options is included later in this chapter.

Conventions Used in this Manual

This manual uses the following conventions:

NOTE

A note contains information relating to the use of this product

CAUTION

A caution contains instructions to avoid damage to the instrument or the equipment connected to it.

WARNING

A warning alerts you to a potential hazard. Failure to adhere to the instructions in a warning could result in personal injury.

Introduction

The most commonly used configuration of the 3172 series is the 3172-W6P2 VXIbus Single-Channel Arbitrary Waveform Generator plus Dual-Channel Pulse Generator. It communicates using the VXI message-based protocol. This high-performance generator combines five powerful instruments in a single C-size card:

The W6 module can be used to generate standard, arbitrary and modulated waveforms and can also be used as a stand-alone counter/timer.

The P2 module has two independent channels that can be used to generate analog-oriented pulse waveforms

The front panel has connectors and indicator lights, but no controls. To control the 3172, use instrument drivers or a soft front panel from your computer.

Supplied with the 3172 is ArbConnection (part number 922336-001), a software application that controls the 3172. ArbConnection allows you to specify, design, or edit waveforms and download them from your computer to the 3172.

The 3172 can be configured to power up in Modern or Legacy mode but is easily programmed between Modern and Legacy modes. See Chapters 3 and 5 for more information.

3172 Configurations (Different Model Numbers)

3172 configuration specifies how the instrument is mechanically constructed and the model number of the final instrument. Figure 1-1 shows the front panels of the various configurations.

The 3172's message-based interface includes LAN and USB. You may communicate with the instrument through the normal VXI controller but take control via the LAN port to completely bypass the backplane interface and control instrument functions and parameters remotely. The USB port is used for memory stick I/O where waveform data can be loaded directly to arbitrary waveform memory. This feature was specifically designed for security reasons where breach of secret waveform data can endanger national security if it falls to the wrong hands. This way, the data is erased immediately as soon as the instrument is turned off but can be restored by a person holding a memory stick with data.

Four types of cards can be installed inside a 3172 module:

- **W6 or W2** – Single channel arbitrary waveform generator
- **P2** – Dual channel pulse generator
- **A3** – 122Vp-p power amplifier

Multiple combinations of these cards can be factory configured for different applications. The following 3172 configurations are available:

- **3172-W6P2** – This model number specifies a single channel arbitrary waveform generator and dual channel pulse generator. In modern mode, this model expands to include all of the features, functions, and operating modes as specified in Appendix A. In legacy mode, this configuration responds to **idn?* with “3171” in the instrument field and is fully compatible with 3171 legacy behavior.
- **3172-W6** – This model number specifies a single-channel arbitrary waveform generator.
- **3172-W2P2** – This model number specifies a single-channel arbitrary waveform generator and dual-channel pulse generator. In modern mode, this model expands to include all of the features, functions, and operating modes as specified in Appendix A. In legacy mode, this configuration responds to *idn? with "3171" in the instrument field and is fully compatible with 3171 legacy behavior except for the fact that the AM is routed differently.
- **3172-P2** –This model number specifies a dual-channel pulse generator.
- **3172-P2P2** –This model number specifies two independent dual channel pulse generators.
- **3172-W6A3** –This model number specifies a single-channel arbitrary waveform generator and a high-voltage power amplifier.

This manual addresses the W6, P2, and A3 functions separately. Information about the W6 also applies to the now deprecated W2 unless stated otherwise throughout this User Manual.

3172-W6 3172-P2 3172-W6P2 3172-W6A3

Figure 1-1, Commonly Used 3172 Configurations

W6 Module Feature Highlights

- Single width, C-size, VXIbus Module
- Provision to generate five types of waveforms: standard, arbitrary, sequenced, modulated, and half-cycle
- 200 MS/s sample clock frequency for generating arbitrary and sequenced waveforms

- Sine and square waveforms to 30 MHz, and other waveforms to over 1 MHz
- Frequency hopping and sweeping
- Modulated waveforms: AM, ASK, FM, FSK, and PSK
- 14-digit sample clock frequency setting, limited by 1 μS/s
- Extremely low phase noise, Clock stability of 1ppm
- PLL function that automatically locks to external signals
- 16-bit vertical resolution; 5 mVp-p to 20 Vp-p into 50 Ω
- Three voltage windows: 0 V to 20 V, -11 V to 11 V, and 0 V to -16 V
- Choice of three matching load impedance: <2 Ω, 50 Ω, and 93 Ω
- 1 M-point memory depth.
- Ultra-fast waveform downloading
- Trigger delay and period-controlled auto re-trigger
- Built-in counter/timer

Figure 1-2, Racal Instruments 3172 (W6P2 Configuration Shown)

P2 Module Feature Highlights

- Fits within a single width, C-size, VXIbus Module
- Two channels, operating independently or synchronously
- 50 MHz clock frequency for generating pulse waveforms
- Independent and fine control of pulse parameters such as period, pulse width and transition times
- Generates single, double and delayed pulses

- Provides normal, inverted and complemented outputs
- Extremely low jitter, Clock stability of 1ppm
- Three voltage windows: Positive, 0 V to 20 V; Symmetrical, -11 V to 11 V; and negative, 0 V to -16 V
- Choice of three matching load impedance: <2 Ω, 50 Ω, and 93 Ω
- Trigger delay and period-controlled auto re-trigger

A3 Module Feature Highlights

- Fits within a single-width, C-size, VXIbus module
- 50 Ω, DC-coupled input
- 122 V_{P-P} output
- 100 kHz output bandwidth
- AWG signals can be fed through the A3 amplifier

ArbConnection Feature Highlights

The ArbConnection software (provided with the 3172) has the following features:

- Virtual control panels
- Arbitrary waveform composer
- Complex pulse composer
- Serial data and FM composers
- Detailed virtual control panels for all functions and modes
- Waveform, modulation, and pulse composers for designing, editing, and downloading complex waveforms
- Automatic detection of active instruments
- Equation editor to generate waveforms from equations
- SCPI command and response editor (simulates ATE operation)
- Translation of waveform coordinates from ASCII and other file formats
- Simplified generation of complex waveform sequences

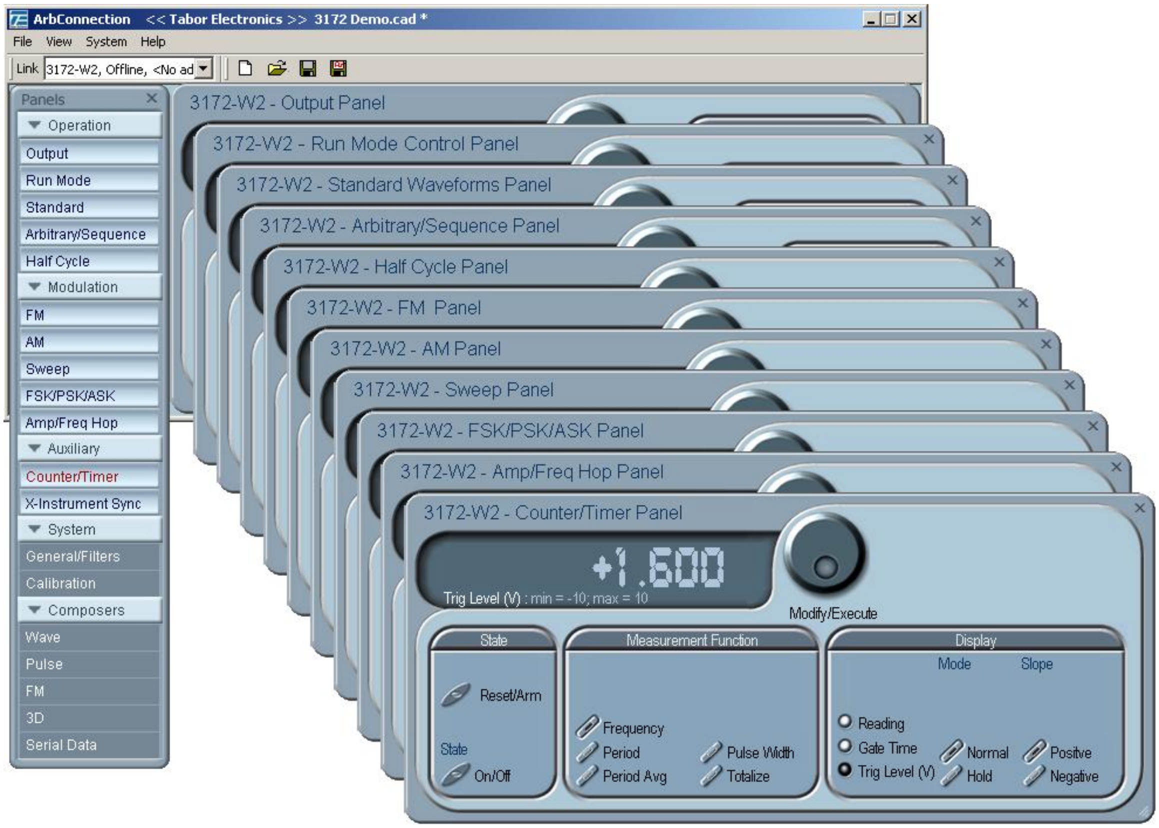

Figure 1-3, ArbConnection Control Panels

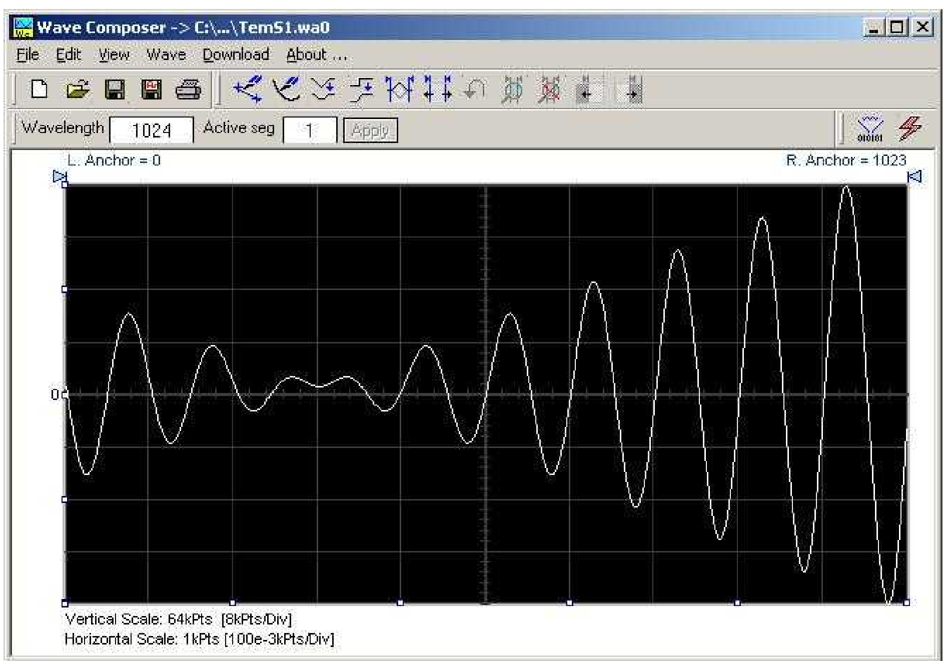

Figure 1-4, ArbConnection Wave Composer Example

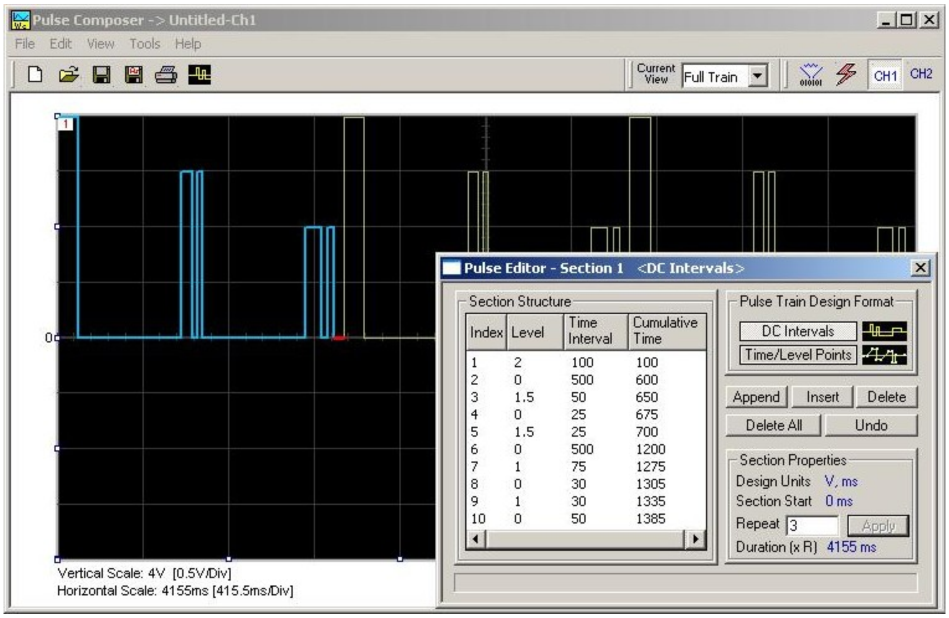

Figure 1-5, ArbConnection Pulse Composer Example

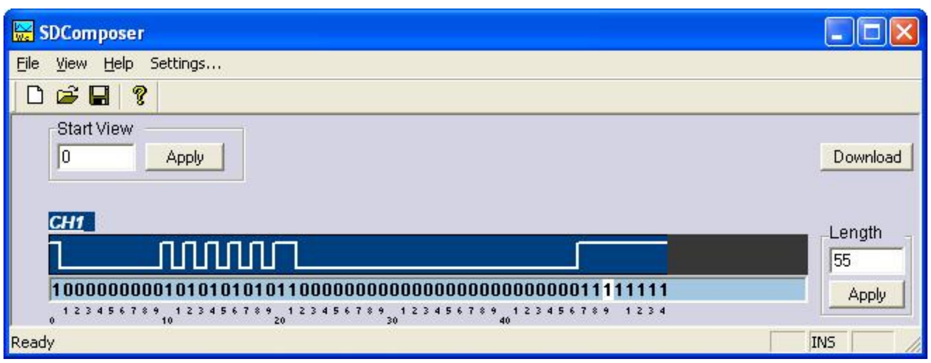

Figure 1-6, ArbConnection Serial Data Composer Example

General Description

This section describes the 3172 general features and performance, as well as its output functions, run modes, and operating modes.

W6 Output Waveforms

The W6 module is a digital waveform generator that creates virtually any type of waveform. Unlike conventional function and pulse generators, the W6 creates waveforms digitally and stores them in memory. A clock generator then clocks the data from memory to a digital to analog converter (DAC) to convert the digital data to an output waveform. Since the waveform memory is volatile, its data is lost when you turn off the instrument. However, you may create as many waveforms as desired on the host computer, and quickly download them to the W6 memory as needed.

The memory size is large enough for most applications. However, you may maximize the effective memory capacity by downloading specific waveforms only when they are required. For example, if one part of your ATE sequence requires a complex waveform that consumes nearly all of the waveform memory, you may delete this waveform after that portion of your sequence is completed, and then quickly download a new waveform for the next part of the sequence.

Depending upon your application, you may be able to change waveform parameters even more quickly without downloading new data. After you download a waveform, you may change the sample clock frequency, amplitude, offset, and run modes without disturbing the downloaded data.

The W6 can divide its memory into smaller segments, and then use these segments to create complex sequences of waveforms.

The W6 can generate the following functions:

- Standard waveforms. The W6 computes these automatically from its built-in equations for sine, square, triangle, and other common waveforms.
- Arbitrary waveforms. Download these to the instrument whenever you need to change from one arbitrary waveform to another.
- Waveform sequences. The W6 builds sequenced waveforms from memory segments that are loaded with the waveforms and are referenced in the sequence table in advance.
- Modulated signals. A direct digital synthesis (DDS) circuit creates these signals without downloading a waveform.
- Half-cycle waveforms. Half-cycle waveforms are similar to standard waveforms, but a programmed delay interval separates the half cycles.
- Phase-Lock Loop (PLL) mode. This special function allows phase locking to an external signal regardless of the wave shape and frequency of that signal. While locked to the external signal, the W6 can generate any of the above functions, with the added control that is attributed to the external signal.
- Counter/timer. The W6 also performs counter/timer measurements.

P2 Output Waveforms

The P2 module is a dual channel pulse generator that generates pulses in an analog manner. It has the capability to generate pulses just as a stand-alone pulse generator does. When using this module, one could program pulse timing parameters in units of time.

All pulse parameters are programmable including period, pulse width, rise and fall times, delay, polarity and more. As a dual channel instrument, you may program different pulse settings for each channel and select if you want the channels to run independently or synchronously.

Depending upon your application, you may be able to change pulse parameters on the fly and without disturbing the downloaded data.

The P2 module can generate the following pulse shapes:

- Single pulse.
- Double pulse.
- Delayed pulse.
- Normal, inverted or pulse complement
- Fixed duty cycle pulse
- Pulse with linear transition

A3 Output Waveforms

The A3 module is a single-channel, high-voltage amplifier. To use it with the W6 waveform generator module, use a coaxial cable to connect the W6 output to the A3 input. You can then command the A3 module to either amplify the W6 signal or simply pass it through to its output connector, bypassing the amplifier.

When the A3 module is set to bypass the W6 output signal, the signal still retains its full bandwidth. However, when the output amplifier is engaged, the amplifier limits the bandwidth to 100 kHz.

Run Modes

The 3172 has two run modes that determine when it will output a waveform. Continuous Run is the basic mode for generating a waveform that does not stop. This mode is appropriate except when the application requires synchronization to external events.

In Interrupted Run mode, the output is either triggered or gated by external signals. The 3172 accepts a trigger event from the front panel connector, a backplane trigger line, or a software command from your computer.

The Run Modes and triggering features are discussed later in this chapter. The run mode options apply to all combinations of W6 and P2 modules.

Frequency Control and Accuracy

An internal reference determines the frequency accuracy of the output waveform. The internal reference provides 1 PPM accuracy and stability over time and temperature. For applications that require better accuracy and stability, or just clock synchronization to external devices, the W6 can use the CLK10 VXIbus signal or the front panel 10MHz REF IN signal (which is available only on a 3172 that has at least one W6 module, i.e., 3172-W6 or 3172-W6P2).

Phase-Lock Loop (PLL)

By activating the Phase-Locked Loop (PLL) function, you may generate any standard or arbitrary waveform while synchronizing with an external signal, or even while tracking it as its frequency changes. This feature is not available on the P2 modules.

Signal Integrity

As technology evolves and new devices are developed each day, faster and more complex signals are needed to simulate and stimulate these new devices. Using the latest technology, the W6 has a bandwidth of 30 MHz, enabling it to accurately duplicate and simulate high frequency test signals. With its outstanding sample clock generator range, 16-bit vertical resolution and its high output voltage amplifier that has a high output bandwidth, one can create mathematical profiles, download sampled waveforms to the instrument and be assured that the waveforms are generated without compromising signal fidelity or system integrity.

Regardless of whether the W6 module, the P2 module or both are used, you can count on the fact that the 3172 utilizes the latest technology to give you the purest and most reliable signals that only a combination of digital and analog instruments can provide.

Frequency Agility

The W6 module has two separate internal clock sources:

- Sample clock generator: The sample clock generator clocks the standard, arbitrary, and sequenced waveforms
- Direct digital synthesis (DDS) circuit: The DDS circuit generates amplitude and frequency related modulation throughout the entire amplitude and frequency ranges of the 3172-W6. This mode is useful for wideband FM, sweep, FSK, PSK, ASK, AM, and frequency hopping, as well as simultaneous frequency, amplitude, and phase modulation (3D modulation).

Amplitude

The output level for both the W6 and P2 modules is programmable from 5 mVp-p to 20 Vp-p into a matched impedance while a programmable offset generator can be programmed to shift the output in either a positive or negative direction or to leave the signal symmetrical about its vertical centerline.

Amplitude Span Ranges

The output span is programmable in three ranges: Positive, from 0 V to 20 V; Symmetrical, from -11 V to 11 V and Negative, from 0 V to -16 V. When range is changed, the peak-to-peak amplitude and offset values automatically default to the factory preset values.

Load Impedance

When programming an amplitude level, it is important that the load impedance matches the source impedance. This is important for a number of reasons, for example, the output calibration is maintained only when a matching impedance is used. In addition, having a matched impedance load assures that the output does not ring out of control. The 3172 has three output impedance options to match industry standard load impedances: <2 Ω, 50 Ω and 93 Ω. The output is calibrated to matching load impedances and, therefore, the selection of the appropriate output impedance is required for precise amplitude level control at the actual load. The consequences of unmatched load impedances is given in Chapter 3.

Remote Control

As with any other VXIbus instrument, the 3172-W6 must be used with a host computer. All of its functions, modes, and parameters are fully programmable using one of the following three ways:

- Low-level programming. Use SCPI commands to program each individual parameter.
- ArbConnection. Use the ArbConnection virtual front panel on the computer screen, which simulates a mechanical front panel. It has push-buttons, displays, and dials to simulate the look and feel of a bench-top instrument.
- Instrument drivers. Use a high-level driver, such as, VXI*plug&play*, IVI or LabVIEW, with your own program to control the 3172.

Safety Considerations

The 3172 has been manufactured according to international CE safety standards – EN-61010. Adjustments, maintenance, or repair of the unit while the covers are removed and power is applied must be carried out only by skilled, authorized personnel. Removal of the covers without authorization shall immediately void the warranty agreement.

Specifications

Appendix A lists the instrument specifications. Specifications apply under the following conditions:

- Output terminated into 50 Ω
- 30-minute warm up period
- Ambient temperature range of 20°C to 30°C.
- For temperatures outside the above temperature range, specifications degrade by 0.1 % per °C.

Options

There are no options available for the 3172 series.

Supplied Accessories

The instrument is supplied with a CD containing the instruction manual, ArbConnection software for Windows® XP, Windows Vista®, and Windows 7 operating systems (part number 922336-001), and VXI*plug&play* soft front panel and drivers (part number 922556).

W6 Front Panel Connectors

The W6 has a single Combo D-sub 24W7 connector, marked as J1, which has seven coaxial connections and 17 low-frequency pins. This connector embeds all of the I/O signal that the W6 generates and needs to control its functions. Figure 1-7 shows the connector and its pin assignments. The following paragraphs describe each Arbitrary Waveform Generator input/output signal.

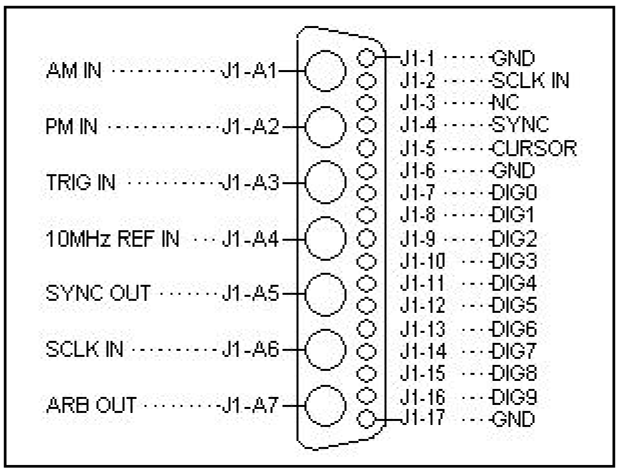

Figure 1-7, W6 Module I/O Connector

AM IN (J1-A1)

The external Modulation coax input provides the capability to externally amplitude modulate the Main Output signal. All standard and arbitrary waveforms can be modulated using this input. Amplitude modulation depth is programmable from 0% to 100%. The AM input is active only after selecting the External AM option. For the 3172-W6, see 3171 AM IN characteristics.

PM IN (J1-A2)

The external Phase Modulation coax input provides the capability to externally modulate the phase of the Main Output signal. The PM input is enabled only when the instrument is placed in PLL operating mode. Phase can be adjusted from -130° to 130° using an amplitude swing of 20 Vp-p.

TRIG IN (J1-A3)

The TRIG IN connector accepts signals that stimulate the generation of output waveforms. The W6 ignores this input when operating in Continuous mode. When placed in Trigger, Gated, or Burst mode, the trigger input is active, and the W6 waits for the proper condition to trigger the instrument. In Trigger and Burst modes, the TRIG input is edge-sensitive, so that a signal transition will trigger the W6. The direction of the transition is programmable. In gated mode, the TRIG signal is level sensitive. The output waveform is enabled when the TRIG signal voltage is beyond a threshold voltage. The threshold voltage and direction are programmable within the range of -10 V to +10 V.

When the PLL function is selected, this input feeds the reference signal to the PLL circuit. The PLL input must be stable and repeatedly cross the trigger level threshold setting. Signals having a low slew rate may cause jitter because of noise. Therefore, make sure that the transition time is fast enough to minimize jitter.

The TRIG IN input is also used in FSK, ASK, and PSK modes where the output shifts between two frequencies, amplitudes, or phases. The output signals have the nominal frequency, amplitude, and phase when the TRIG IN level is at logic 0, and a shifted frequency, amplitude, or phase when the TRIG IN level is at logic 1.

10 MHz REF IN (J1-A4)

This input accepts a 10 MHz reference signal. At the factory, this input is configured for TTL logic levels. It may be changed to 0 dBm, but only by qualified service personnel.

The EXT 10 MHz input is available for applications requiring a more accurate, stable reference than can be attained by the 1 ppm TCXO reference built into the W6 module. The reference input is active only after selecting the External 10 MHz Reference Source option.

SYNC OUT (J1-A5)

The SYNC OUT connector outputs a single TTL-level pulse for synchronizing other instruments, such as an oscilloscope, to the output waveform. The SYNC signal always appears at a fixed point relative to the waveform. The location of the SYNC signal relative to the waveform is programmable, as is the pulse width. When the sweep or any other modulation function is enabled, the SYNC connector is also useful as a marker output. For digital patterns the SYNC output is controlled by bit M12 (J1-4) of the 16-bit digital word.

SCLK IN (J1-A6)

The EXT SCLK connector accepts sample clock signals from an external source. It is AC-coupled to accommodate positive ECL (PECL) or negative ECL (NECL) amplitude level clock signals having frequencies from DC to 200 MHz. This signal replaces the internal clock generator, either for low-noise applications or for synchronization purposes. The sample clock input is active only after selecting the External Sample Clock Source option.

ARB OUT (J1-A7)

The main output coax pin (J1-A7) provides standard, Arbitrary and modulated Waveforms. The output impedance of this output is selectable from <2 Ω, 50 Ω, or 93 Ω. The cable connected to this output should be terminated with a 50 Ω or 93 Ω resistance. The output amplitude is specified when connected to a 50 Ω load. If the output is connected to a different load resistance, determine the actual amplitude from the resistance ratio of the internal output impedance to the load impedance

SCLK IN (J1-2)

The SCLK IN connector is connected for compatibility with the 3171 legacy hardware. This pin is not connected in the 3172-W6 configuration. It is AC-coupled to accommodate positive ECL (PECL) or negative ECL (NECL) amplitude level clock signals having frequencies from DC to 40 MHz. This signal replaces the internal clock generator, either for low-noise applications or for synchronization purposes. The sample clock input is active only after selecting the External Sample Clock Source option.

Digital Pattern Outputs

When enabled by the Digital Stimulus Pattern Command, the below listed pins output TTL levels at the programmed sample rate. Up to 16,000 different digital patterns can be stored in the waveform memory. The maximum depth of the patterns is the same as for individual arbitrary waveform segments. When the Digital Pattern output pins are enabled the main output is disabled. The list below describes the function of each of the J1 pins.

J1-1, J1-6, and J1-17 – Ground connections

J1-3 – Not used

J1-4 – SYNC Output. TTL level signal used during digital pattern generation. SYNC out is controlled by bit 12 of the 16-bit digital word.

J1-5 – Cursor Output. TTL level signal used during digital pattern generation. Cursor out is controlled by bit 13 of the 16-bit digital word.

J1-7 – Bit 0 of the 16-bit digital word.

J1-8 – Bit 1 of the 16-bit digital word.

J1-9 – Bit 2 of the 16-bit digital word.

J1-10 – Bit 3 of the 16-bit digital word.

J1-11 – Bit 4 of the 16-bit digital word.

J1-12 – Bit 5 of the 16-bit digital word.

J1-13 – Bit 6 of the 16-bit digital word.

J1-14 – Bit 7 of the 16-bit digital word.

J1-15 – Bit 8 of the 16-bit digital word.

J1-16 – Bit 9 of the 16-bit digital word.

P2 Front Panel Connectors

The P2 has a single Combo D-sub 5W5 connector, marked as J2, which has five coaxial connections. This connector embeds all I/O signals that the P2 generates and needs to control its functions. Figure 1-8 shows the connector and its pin assignments. The following paragraphs describe each Pulse Generator input/output signal.

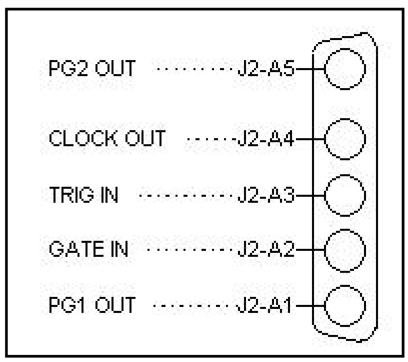

Figure 1-8, P2 Module I/O Connector

PG1 OUT (J2-A1)

The main output coax pin (J2-A1) provides a path for channel 1 pulse waveforms. The source impedance of this output is selectable from <2 Ω, 50 Ω, or 93 Ω. The cable connected to this output should be terminated with a 50 Ω or 93 Ω resistance. The output amplitude is specified when connected to a 50 Ω load. If the output is connected to a different load resistance, determine the actual amplitude from the resistance ratio of the internal output impedance to the load impedance

GATE IN (J2-A2)

The GATE IN connector accepts gating signals that stimulate the generation of output waveforms. It can be programmed to gate either channel 1 or channel 2. The P2 module ignores this input when operating in Continuous mode. When placed in Gated run mode the gate input is made active, and the P2 module waits for the proper condition to gate the instrument. The gate input is level -

sensitive, so that crossing a predetermined signal level will gate one of the outputs. The polarity of the gate is programmable. The input is a TTL level signal and is pulled up to +5 V through a 4.7 kΩ resistor.

TRIG IN (J2-A3)

The TRIG IN connector accepts trigger events that stimulate the generation of output waveforms. It can be programmed to stimulate either channel 1 or channel 2. The P2 module ignores this input when operating in Continuous mode. When placed in Trigger, or Burst mode, the trigger input is made active, and the P2 input waits for the proper event to trigger the instrument. In Trigger and Burst modes, the TRIG input is edge-sensitive, so that a signal transition will trigger one of the P2 outputs. The output waveform is triggered when the TRIG signal voltage transitions through a threshold voltage. The input is a TTL level signal and is pulled up to +5 V through a 4.7 kΩ resistor.

CLOCK OUT (J2-A4)

The CLOCK OUT connector outputs a fixed level TTL signal capable of driving a 50 Ω load to a level > 3 Volts. The signal is derived directly from either channel 1 or from channel 2 periods.

PG2 OUT (J2-A5)

The main output coax pin (J2-A5) provides a path for channel 2 pulse waveforms. The source impedance of this output is selectable from <2 Ω, 50 Ω, or 93 Ω. The cable connected to this output should be terminated with a 50 Ω or 93 Ω resistance. The output amplitude is specified when connected to a 50 Ω load. If the output is connected to a different load resistance, determine the actual amplitude from the resistance ratio of the internal output impedance to the load impedance

EXT WID

The external pulse width input is using the external signal to shape and define the width of the output pulses. The input is a TTL level signal and is pulled up to +5 V through a 4.7 kΩ resistor.

As long as the input remains low, the output idles on its low level setting. When the input goes high, the output generates the high level and the width is proportional to the time that the input remains high level.

Output Waveforms

The W6 module can generate five types of waveforms:

- Standard (Fixed)

- Arbitrary
- Sequenced
- Modulated
- Half-cycle

The **P2** module can generate a variety of pulses through two independent outputs:

- Single
- Double
- Delayed

W6 - Standard (FIXED) Waveforms

The W6 must pre-load its memory before it can generate standard waveforms. On power-up, the waveform memory contains no specific data. The sine waveform, which is the default, is computed and loaded into the waveform memory as part of the reset procedure. Later, if you select another standard waveform, the 3172-W6 computes the waveform points and loads them into the waveform memory.

Every time the user selects a new waveform, there is some delay for the processor to compute the data and download it to memory. The delay interval depends on the complexity of the waveform and the number of points the processor has to calculate. It is good practice to add sufficient delay to a test program to allow for this delay. The delay could range from a few milliseconds to a few seconds, and there are no special rules beside trial and error to determine the necessary delay time.

Nine standard waveform shapes are available:

- Sine
- Triangle
- Square
- Pulse/Ramp
- Sine(x)/x
- Pulse
- Gaussian Pulse
- Rising/decaying Exponential Pulse
- Noise
- DC

Each waveform has parameters for modifying it to suit your requirements.

W6 - Arbitrary (User) Waveforms

The waveform memory can store one or more arbitrary, or user-defined, waveforms. The regular W6 configuration is supplied with 1 megasample.

You may allocate the entire memory for a single waveform or you may divide the memory into smaller segments and load each segment with a different waveform. By dividing the memory into multiple segments, you may program the instrument to output the waveform one segment at a time, using a simple command each time you want to select a different memory segment. There are no limitations on the shape of the arbitrary waveform as long as it meets certain criteria such as minimum and maximum lengths and does not exceed the dynamic range of the DAC (65,535 counts).

W6 - Sequenced Waveforms

The sequence generator lets you link and loop segments in any order. For a simple example of a sequenced waveform, see Figures 1-9 through 1-12. The waveforms in Figures 1-9 through 1-11 are placed in memory segments 1, 2, and 3, respectively. The sequence generator links and loops these waveforms in a predefined order to generate the waveform shown in Figure 1-12.

The sequence circuit is useful for generating long waveforms with repeated sections. Although the waveform only needs to be programmed once, the sequencer loops on this segment as many times as selected. When in sequenced mode, there is no time delay between linked or looped segments.

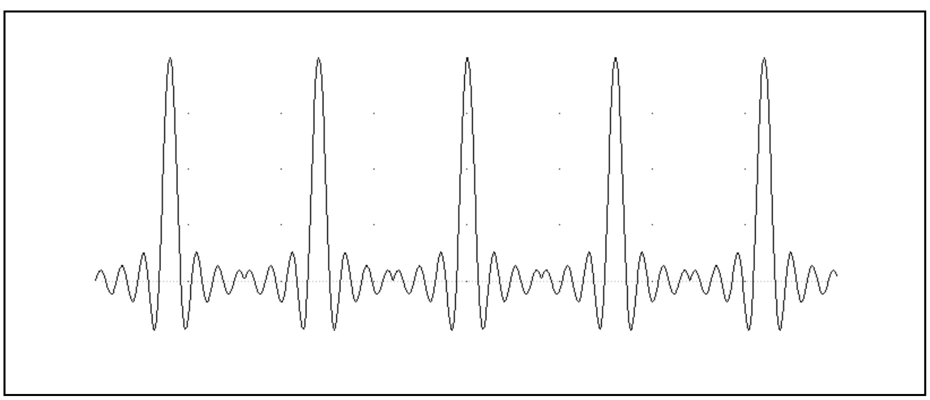

Figure 1-9, Segment 1 – Sine (x)/x Waveform

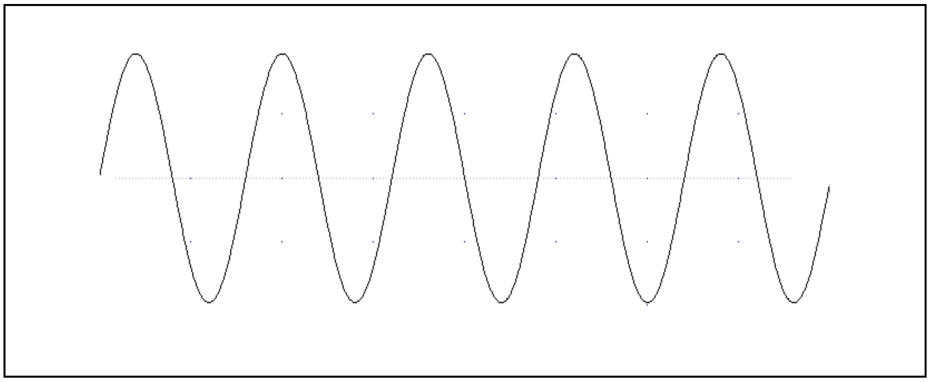

Figure 1-10, Segment 2 – Sine Waveform

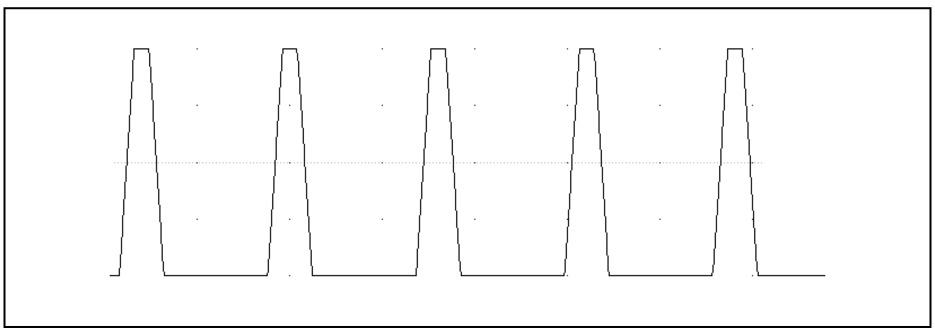

Figure 1-11, Segment 3 – Pulse Waveform

The following sequence was made of segment 2 repeated twice, segment 1 repeated four times, and segment 3 repeated two times.

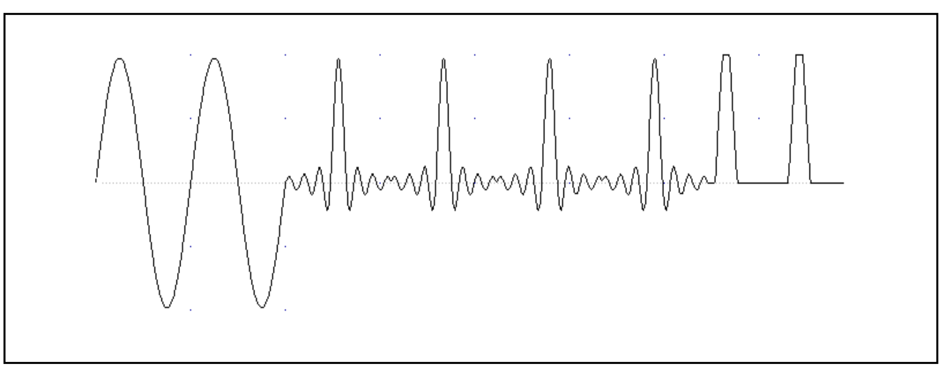

Figure 1-12, Sequenced Waveforms

W6 - Modulated Waveforms

The use of direct digital synthesis (DDS) technology gives the W6 frequency agility. During operations such as sweep, FSK, FM, and other modulation modes, the W6 quickly synthesizes the modulated waveform using the DDS circuit. The varieties of modulated waveforms available are described below.

Sweep

The W6 can sweep the output frequency between minimum and maximum values that you specify. You may sweep up or down using linear or logarithmic increments. Sweep frequency is programmable from 10 Hz to 30 MHz, and sweep times can range from 1.4 μs to 40 seconds. Sweep mode is compatible with Continuous, Triggered, and Gated modes.

Sweep modes with triangle and square waveforms are computed and placed in memory as complete waveforms. This adds delay before the initial output is available while the software computes the waveform. All sine swaps use the DDS circuit, thus no computation time is required.

FM

The FM function modulates the frequency of the W6 output waveform. You can modulate the output using built-in standard or arbitrary waveforms. FM is available in Continuous, Triggered, and Gated modes.

The W6 generates two types of frequency modulation: standard and arbitrary. For standard modulation, the modulation waveform is selected from a built-in library of four standard waveforms: sine, triangle, square, and ramp. For arbitrary modulation, complex modulation signals are loaded into modulation waveform memory. There are 10,000 points allocated specifically for modulation waveform memory.

AM

The internal AM function modulates the amplitude of the W6 output waveform. Four standard modulating waveforms are available: sine, triangle, square, and ramp. AM can be used in Continuous, Triggered, and Gated modes. Modulation depth is programmable from 0% to 100% and up to 200% in some cases.

Frequency Hopping

The Frequency Hopping function causes the output frequency to hop through a sequence of frequencies. The amount of time the W6 dwells on each frequency is programmable. You may opt to set the dwell time uniformly over the entire hop list.

The frequency hop table can contain up to 1,000 frequency values ranging from 10 Hz to 30 MHz.

Amplitude Hopping

The amplitude hopping function causes the output amplitude to hop through an amplitude list. The amount of time the 3172-W6 dwells on an amplitude level is programmable for each hop. You may also set the dwell time uniformly over the entire hop list.

The amplitude hop table contains up to 5,000 different amplitude values ranging from 0 V to 11 V.

FSK

FSK (frequency shift keying) shifts the output between two frequencies. The logic level of the TRIG/PLL input determines the instantaneous frequency value. When the trigger slope is set to positive and the TRIG/PLL is false, the output is at the base frequency. When TRIG/PLL is true, the output frequency is shifted by an offset. To reverse the trigger polarity, select the negative trigger slope.

PSK

PSK (phase shift keying) shifts the phase of the output between 0° and 180°. The logic level of the TRIG/PLL determines the phase value. When the trigger slope is set to positive and the TRIG/PLL is false, the phase shift is 0°. When TRIG/PLL is true, the phase shift is 180°. To reverse the trigger polarity, select the negative trigger slope.

ASK

ASK (amplitude shift keying) shifts the output between two amplitudes. The logic level of the TRIG/PLL input determines the instantaneous amplitude value. When the trigger slope is set to positive and the TRIG/PLL is false, the output is at the base amplitude. When TRIG/PLL is true, the output amplitude is shifted by an offset. To reverse the trigger polarity, select the negative trigger slope.

W6 - Half Cycle Waveforms

The W6 generates three types of half-cycle waveforms: sine, triangle, and square. The frequency range is 10 mHz to 1 MHz, and the delay between half cycles is programmable from 100 ns to 20 seconds in increments of 20 ns. You may also program the starting phase of the waveforms from 0.1° to 359.9°.

W6 - Counter/ Timer

The W6 can operate as a counter/timer to measure frequency, period, averaged period, and pulse width, and to count events. As a counter/timer, it measures frequency to over 100 MHz with gate times of 100 µs to 1 s. When using a gate period of one second, it provides seven digits of resolution with an initial accuracy of 1 ppm.

P2 – Pulse Waveforms

The P2 has two output channels of which each can operate independently of the other but both can also be locked out to operate in synchronized mode. Each channel can output the following pulse shapes: normal, double and delayed. Pulse polarity can be selected from: normal, Inverted and complemented and transitions can be defined to be fast or linear. The 3172 run modes apply to the pulse output and hence the pulse can be triggered, gated, or generate a burst of pulse trains.

The pulse waveform is generated using analog circuits but the controlling circuits are digital and therefore, the parameters can be programmed to a great degree of accuracy while the analog circuits guarantee for pulse stability and signal integrity.

All pulse parameters are controlled using dedicated remote commands however, an additional input at the front panel provides an option of external control of the pulse width. When selected, an external signal modifies the width of the pulse when it transitions through certain threshold levels.

Each of the two pulse outputs has its own clock source which allows each channel to operate in an entirely different rate and mode from the other channel. The P2 module has provisions to lock the two channels together so they run from a single clock source and the 3172 has provisions to run all W6 and P2 outputs from a single clock generator so that all 3172 outputs, regardless of whether they are W6 or P2 channels, can be synchronized to the same source.

Detailed information on pulse parameters and control options is given in Chapter 3.

General Run Modes

The 3172 may operate in one of four run modes: Continuous, Triggered, Gated, and Burst. These modes are described below. Note that the W6 behaves differently when it generates modulated waveforms. The description below applies to standard, arbitrary, and sequenced waveforms. The Modulation mode is described later.

The 3172 responds to a variety of trigger sources: front panel triggers, the TRIG IN connector, VXIbus backplane trigger lines

(TTLTrg0-7), and software triggers from the computer. There are also two built-in trigger generators. One repeats itself at pre-programmed intervals from 100 ns to 20 seconds. The other has a programmable delay. The re-trigger delay is measured from the end of a signal to the start of the next signal. You may program the re-trigger delay from 100 ns to 20 seconds, in increments of 20 ns.

Continuous Mode

In Continuous mode, the 3172 generates the selected waveform continuously at the selected frequency, amplitude, and offset. The generator will begin waveform generation as soon as the waveform and its parameters have been programmed, and will stop only when turned off or placed in one of the interrupted run modes.

Triggered Mode

In Triggered mode, you may program the trigger circuit to respond to positive or negative transitions of the trigger input signal. When triggered, the generator outputs one waveform cycle, and then remains idle at an amplitude level equal to the voltage of the first point of the waveform. You may set the instrument to receive triggers from the front panel connector, backplane, or the trigger command in your software.

A re-trigger circuit requires only one trigger event, after which it automatically generates a series of triggers. In this case, the re-trigger delay parameter determines the time between waveform cycles.

The trigger signal, whether it comes from the front panel, VXIbus trigger line, or a software command, has to pass through electrical circuits. These circuits cause a small delay known as system delay. This delay determines the amount of time it will take from a valid trigger edge to the moment that the output reacts. System delay cannot be eliminated completely, and must be accounted for when using a trigger signal.

Note that the W6 modules have a single input for trigger and gated signals but the P2 module has two inputs, one for trigger and the other for gate. On the P2 module, you first need to associate the trigger and gate inputs with the appropriate channel because there is only one input available for both.

Gated Mode

In Gated mode, the 3172 circuits generate an output waveform as long as a gating signal is present. The instrument can be programmed to gate on two different signal types. The normal mode is level sensitive, where the output is enabled only while the trigger signal is above the trigger level threshold voltage. The second mode is transition (edge) sensitive, where the gate opens on the first transition and closes on a subsequent transition.

Regardless of the selected gating mode, the generator always completes the waveform at the end of the gate and then idles at a DC level.

Note that the W6 modules have a single input for trigger and gated signals but the P2 module has two inputs, one for trigger and the other for gate. On the P2 module, you first need to associate the trigger and gate inputs with the appropriate channel because there is only one input available for both.

Burst Mode

The Burst mode is an extension of the Triggered mode where the generator is programmed to output a pre-determined number of waveforms. The sources to trigger a burst are the same as for the Triggered mode.

Trigger Sources

The 3172 responds to a variety of trigger sources such as the TRIG IN connector, backplane trigger lines (TTLTrg0-7), and a software trigger. There are also two built-in, self-generating trigger generators. One repeats itself at pre-programmed intervals from 100 μs to 20 seconds. The other has a programmable delay time. The re-trigger delay is measured from the end of the signal to the start of the next signal and programmed from 100 ns to 20 seconds with a resolution of 20 ns.

Modulation Run Modes (W6 only)

As previously mentioned, the 3172 has four run modes: Continuous, Triggered, Gated, and Burst. However, the W6 behaves differently when generating modulated waveforms.

While the modulated and non-modulated run modes are similar, the modulated run mode offers two start options for the output signal during idle. Idle is the period of time before the output is triggered or gated to generate a modulated waveform.

The first option is where the W6, before receiving a trigger event, outputs continuous, non-modulated waveforms. When the trigger or gate occurs, the W6 outputs the modulated waveform. When that waveform has completed, the instrument resumes outputting non-modulated waveforms.

The second option is where the W6, before receiving a trigger or gate, outputs a DC level. When triggered or gated, the W6 outputs the modulated waveform. When that waveform has completed, the instrument resumes outputting a DC level.

Synchronization of

A single or dual waveform generator in a single slot, no matter how advanced, may become a limiting factor for applications requiring

Multiple 3172-W6 Modules

multiple simultaneous waveforms.

Various techniques exist to synchronize the outputs of multiple waveform generators, but none are simple because real synchronization requires sharing of the reference and sample clocks, as well as signals that control the starting phase of the waveform. The 3172 can use the VXIbus Local Bus (LBUS0-7) to synchronize adjacent modules. The Local Bus lines are short and can tolerate high-frequency signals, but modules must be placed in the VXIbus chassis in a fixed master/slave configuration, and local bus jumpers must be installed. Details are provided in Chapter 2 in the Local Bus Configuration section..

In either case, the slave instrument(s) are locked to the frequency and starting phase of the master module. After lock has been achieved, the starting phase of the slave modules may be shifted with respect to the master module to create a multi-phase system. The starting phase is programmable from 0° to 360°.

PLL Synchronization (W6 only)

PLL synchronization is another technique for synchronizing multiple waveform generators. In this case, synchronization is not between pairs of 3172-W6 modules, but between the 3172-W6 and any external device that generates signals stable enough to satisfy the PLL input requirements. When placed in this mode, the 3172-W6 measures the frequency stability of the input signal and determines whether or not it is valid. Then, the built-in counter/timer circuit measures the frequency of the signal and centers its lock-in range on this frequency. The 3172-W6 thus locks automatically onto the frequency of the external signal. Note that there is no need for manual initiation or operator intervention for the PLL function to find and lock onto an external reference. The PLL range is 500 Hz to 10 MHz.

After phase locking has been established, the start phase of the 3172-W6 waveform can be shifted with respect to the start phase of the external reference within the range of -180° to +180°, with phase increments as low as 0.01° (fine phase control). The reference signal is applied to the front-panel TRIG IN connector. In this way, the same reference can be applied to multiple modules to generate multi-phase signal patterns. The TRIG IN input has a programmable trigger level and programmable slope.

Phase Modulation (W6 only)

When the 3172-W6 is placed in PLL mode, there are two ways to control the phase offset. The first way is to modify the phase offset setting using SCPI commands. This method is quite accurate, and allows phase offset adjustments in increments of 0.01°.

The second way is to apply a voltage to the PM IN port of the 3172-W6. This changes the start phase of the synthesizer, proportional to the voltage level at the phase modulation input. An input voltage of 1 V modifies the phase by 20°. The phase changes between -130° and 130° as the applied voltage goes from -10 V to 10 V. The instrument responds to AC changes on the PM IN port throughout the frequency range of DC to 10 kHz, but the time to lock depends on the lock frequency and the number of waveform samples.

Filters

The W6 has two elliptic filters (60 MHz and 120 MHz) and two Bessel filters (2 MHz and 20 MHz). You may switch these filters in to reduce harmonics or high frequency spurs.

Access to the elliptic filters is disabled while the W6 is generating standard sine waveforms because these filters are used automatically to construct optimal sine waveforms.

Output State

For safety reasons, the 3172 powers up with its output turned off. In fact, the output circuit is disconnected from the output connector using a mechanical relay, which eliminates erroneous and uncontrolled transitions that may occur during power-up. This protects equipment that remains connected to the output when the mains power fails or the system is powered down.

Mechanical relays have a settling time of about 2 ms. Therefore, when writing software, allow enough time for the relay to close before using the signal at the output connector.

Programming the 3172-W6

The 3172 has no front panel control; therefore, you must use a computer to communicate with the instrument. There are a number of ways to do this including the use of an appropriate software driver on the host computer. The specifics of communication are discussed in later chapters.

An alternative to a driver is to use the SCPI (Standard Commands for Programmable Instruments) language. Chapter 5 explains the details of SCPI programming.

The ArbConnection software application is supplied with the 3172. ArbConnection provides a user interface that allows you to control the 3172 interactively. Chapter 4 provides detailed instructions for using ArbConnection.

This page was intentionally left blank.

Chapter 2
Installation

Preparation for Use

Preparation for use includes selecting the required logical address, configuring the Local Bus (if required), and installing the module in a VXIbus chassis.

Logical Address Selection

The VXIbus Chassis Resource Manager identifies a module in the system by the module’s address. VXIbus logical addresses can range from 0 to 255. However, logical address 0 is reserved for the Resource Manager. Addresses 1 to 254 are reserved for VXIbus modules. Logical address 255 permits the Resource Manager to dynamically configure the module logical address.

To change the 3172 logical address, use the 8-segment DIP switch (S1) accessible from the side of the module near the rear of the case. Figure 2-1 shows the logical address switch. The switch segments are marked with numbers 1 to 8. Each switch segment represents a binary digit of the 8-bit binary logical address. The segment marked “1” represents the least-significant digit. A switch is active (equal to 1) when it is moved downward to the “ON” position.

Astronics Test Systems ships the 3172 with the logical address set to 2 as shown in Figure 2-1.

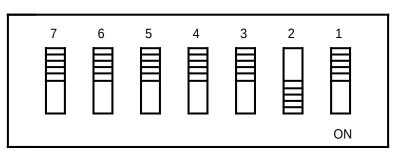

Figure 2-1, Switch S1 (Set to Logical Address 2)

Local Bus Configuration

A 3172 can synchronize (phase lock) with other 3172 modules installed in the same VXI chassis. One of the synchronization methods uses the VXI Local Bus lines (LBUS0-LBUS7). The Local Bus has advantages of high bandwidth and the capability to synchronize any quantity of modules in the same chassis.

To use the Local Bus, the modules to be synchronized must be configured as instructed below, and must be operated in Modern Mode (not Legacy Mode).

NOTE
Originally, configuration of Model 3172 for Local Bus operation was performed at the factory. In September 2013, a feature was added to allow the user to configure the 3172. If you need to configure a 3172 module for Local Bus operation, and it does not have the side-panel access openings shown in Figure 2-2, contact Customer Support. Contact information is provided in the Warranty Statement at the front of this manual.

When synchronizing (phase locking) a group of modules, they must be installed in a contiguous group of slots in the VXIbus chassis. The reference module (master) that the others will synchronize to must be immediately to the left of the other modules (slaves) in the group.

To enable use of the Local Bus, you must install Local Bus jumpers (supplied with 3172) into each module. Figure 2-2 shows the installation locations for the jumpers, which you will install onto the internal printed circuit board through the openings in the side panel. Each jumper slides onto two header pins, connecting them together. It may be helpful to use needle-nose pliers to install the jumpers.

The specific jumpers required for each 3172 depend on its position within the synchronized group:

1. The first module on the left (master) requires jumpers LK15 and LK16. Do not install LK13 or LK14 on this module.
2. The last module on the right requires jumpers LK13 and LK14. Do not install LK15 or LK16.
3. If there are modules between the first and last modules, they each require jumpers LK13, LK14, LK15, and LK16.

As mentioned above, the 3172 must be in Modern Mode (not Legacy Mode) to use the Local Bus for synchronizing modules. To place the 3172 in Modern Mode, send the following SCPI command:

FORMat:INSTrument MODern

You may abbreviate the command by omitting all lower-case characters. When power is cycled, the 3172 defaults to Legacy Mode, even if it was in Modern Mode while power was being turned off.

Commands used for controlling local bus synchronization are detailed in Chapter 5 in the Instrument & Output Control Commands section (see INSTrument:COUPle commands).

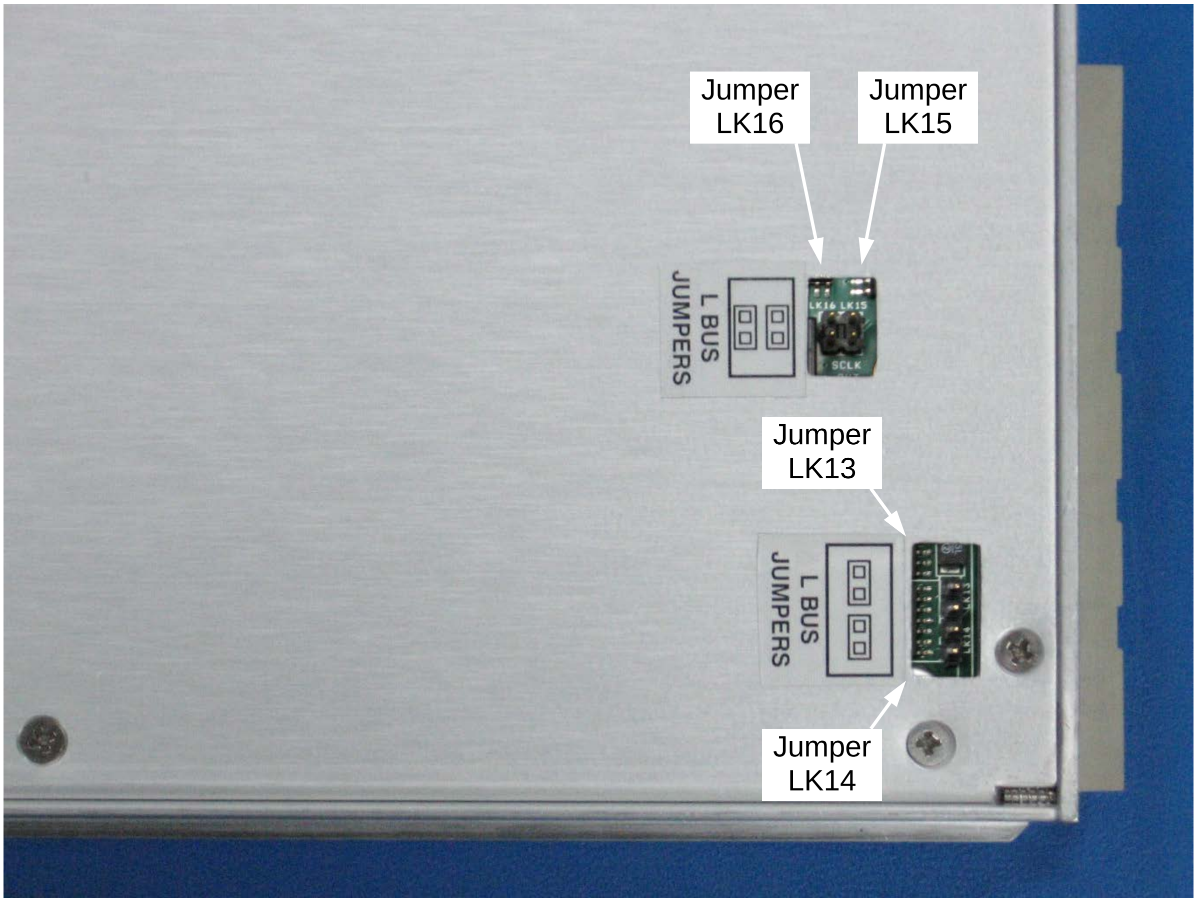

Figure 2-2, Local Bus (LBUS) Jumper Installation

Installation

If the 3172 is not configured to use the Local Bus, you may install it into any empty slot in the VXIbus chassis except slot 0. If the 3172 is configured for Local Bus operation, then take care to comply with the configuration rules from the previous section.

When inserting the instrument into the mainframe, gently rock it back and forth to seat the connectors into the backplane receptacle. The ejectors will be at right angles to the front panel when the instrument is properly seated into the backplane. Use the captive screws above and below the card ejector handles to secure the instrument into the chassis.

Controlling the Instrument from a Remote Device

In general, the 3172 is controlled from a remote device using the VXIbus slot 0 controller. In addition, when configured in a 3100M carrier, there is an additional front panel LAN connector that allows communication with a web page (LXI specification) and USB port. This allows you to transfer data directly to the waveform memory.

To communicate with the 3172 you may either write your own software or use one of the soft front panel programs that allow access to all instrument modes, functions, and parameters. If you wish to use ArbConnection, insert the supplied CD and follow the instructions on the screen to install the program. You may also use the soft panels that are supplied with the VXIplug&play drivers on the same CD. 3172 users that intend to control the instrument from a web page must program the LAN parameters before it can be used on the network. Information on how to program the LAN parameters is given below. Note that as long as the instrument is powered on, the LAN parameters will not change. For a new LAN setting to take effect, turn the power off and then back on.

Installing Software Utilities

The 3172 is supplied with a CD containing ArbConnection and the VXI*plug&play* driver, and possibly an IVI compatible driver for use with the Ethernet interface. It also includes the user manual. You should store the CD in a safe place in case you need to restore damaged files or load the software onto different computers. The latest drivers, and firmware are available for download from the Astronics Test Systems at http://www.astronicstestsystems.com/support/downloads.

ArbConnection lets you control instrument functions and features from a remote computer. It also lets you generate and edit arbitrary waveforms, sequence tables, and modulated signals, and then download these to the 3172.

You may use ArbConnection to control the 3172 without writing software. However, for maximum flexibility, you may control the 3172 at a low level using SCPI commands in your own software. Note that for register-based models, SCPI commands need to be directed to a DLL instead of to the VISA library. In either case, you may use SCPI commands through ArbConnection's command editor without programming.

Chapter 4 provides installation and operating instructions for ArbConnection.

Connecting to a LAN Network

The 3172 has a front panel connector that allows connection to a local area network system. This LAN port has three purposes:

Download waveform data directly from an external computer without using the VXIbus controller.

Control the 3172 in a system that does not have a VXIbus slot 0 controller.

Use Ethernet to control VXIbus modules adjacent to the 3172 using VXIbus local bus lines (for modules designed to interface with the 3172 in this way).

The programming section of this manual lists the default settings. Additional descriptions of LAN settings are given below.

Direct connection between a single host computer and a single device is also possible, but you must use a special cable that has it's transmit and receive lines crossed. If your site is already wired, connect the 3172 via twisted pair Ethernet cable. Take care that you use twisted pair wires designed for 10/100 BaseT network use (phone cables will not work). Refer interconnection issues to your network administrator. After you connect the 3172 to the LAN port, proceed to the LAN Configuration section in this chapter for instructions how to set up LAN parameters.

LAN Configuration

There are several parameters that you may have to set to establish network communications with a LAN interface. Primarily you'll need to establish an IP address. You may need to contact your network administrator for help in establishing communications with the LAN interface. To change LAN configuration, you'll need to use some LAN commands that are listed in the programming reference. The programmed parameters will be updated with the new setting only after you turn the VXI chassis off and on once.

Note there are some LAN parameters cannot be accessed or modified; These are: Physical Address and Host Name. These parameters are set in the factory and are unique for this product. The only parameters that can be modified are the IP Address, the Subnet mask and the Default gateway. Correct setting of these parameters is essential for correct interfacing with the LAN network.

Description of the LAN settings is given in the following. Information how to modify the LAN setting is given in the programming section of this manual.

Note

Configuring your LAN setting does not automatically select the LAN as your active remote interface.

There are three LAN parameters that can be modified and adjusted specifically to match your network setting; These are described below. Consult your network administrator for the setting that will best suit your application.

- **IP address** - The unique, computer-readable address of a device on your network. An IP address typically is represented as four decimal numbers separated by periods (for example, 192.160.0.233). Refer to the next section - Choosing a Static IP Address.
- **Subnet mask** - A code that helps the network device determine whether another device is on the same network or a different network.
- **Gateway IP** - The IP address of a device that acts as a gateway, which is a connection between two networks. If your network does not have a gateway, set this parameter to 0.0.0.0.

Choosing a Static IP Address

For a Network Administered by a Network Administrator

If you are adding the Ethernet device to an existing Ethernet network, you must choose IP addresses carefully. Contact your network administrator to obtain an appropriate static IP address for your Ethernet device. Also have the network administrator assign the proper subnet mask and gateway IP.

For a Network without a Network Administrator

If you are assembling your own small Ethernet network, you can choose your own IP addresses. The format of the IP addresses is determined by the subnet mask. You should use the same subnet mask as the computer you are using with your Ethernet device. If your subnet mask is 255.255.255.0, the first three numbers in every IP address on the network must be the same. If your subnet mask is 255.255.0.0, only the first two numbers in the IP addresses on the network must match.

For either subnet mask, numbers between 1 and 254 are valid choices for the last number of the IP address. Numbers between 0 and 255 are valid for the third number of the IP address, but this number must be the same as other devices on your network if your subnet mask is 255.255.255.0.

Table 2-1 shows examples of valid and invalid IP addresses for a network using subnet mask 255.255.255.0. All valid IP addresses contain the same first three numbers. The IP addresses in this table are for example purposes only. If you are setting up your own network, you probably do not have a gateway, so you should set these values to 0.0.0.0.

Table 2-1, Valid and Invalid IP Addresses for Subnet Mask 255.255.255.0

IP Address	Comment
123.234.45.211	Valid.
123.234.45.213	Valid. The first three numbers match the previous IP address. The fourth number must be a unique number in the range of 1 to 254.
123.202.45.214	Invalid. Second number does not match the previous IP addresses. The first three numbers must match on all IP addresses with subnet mask 255.255.255.0.
123.234.45.0	Invalid. The first three numbers are valid but the fourth number cannot be 0.
123.234.45.255	Invalid. The first three numbers are valid but the fourth number cannot be 255.

TIP

To find out the network settings for your computer, perform the following steps:

- **For Windows XP/Vista/7**

  1. **Open a DOS prompt.**
  2. **Type IPCONFIG.**
  3. **Press <Enter>.**

If you need more information, you can run ipconfig with the /all option by typing IPCONFIG /all at the DOS prompt. This shows you all of the settings for the computer. Make sure you use the settings for the LAN adapter you are using to communicate with the LAN device.

Select the Ethernet adapters you are using to communicate with the *Ethernet device* from the drop-down list.

Connecting to the USB Port

The 3172 has a front panel USB connector that allows connection to a USB memory device. This USB port has only one purpose, to allow the downloading of waveform data directly from a USB memory device without going through the VXI controller. The waveform data has to be stored on the USB memory device in a special format. The negotiation between the USB memory device and the instrument is automatic. The application program can then select the loaded memory segment for use in an application.

This page was intentionally left blank.

Chapter 3
Operation

Overview

This chapter explains how to operate the 3172. Unlike a bench-top instrument, the 3172 requires a computer to turn on functions, change parameters, and configure various operating modes. Two software applications are available to control the instrument: VXI*plug&play* soft front panels (SFPs) and ArbConnection. For the experienced programmer, VXI*plug&play* drivers and a set of SCPI commands are available.

NOTE

The Model 3172 can be fitted with W2 (deprecated), W6, P2, and A3 modules. Throughout the following discussion, where the model number 3172 is cited, the description is common to both W6 and P2 modules. When operation differs between W6 and P2, the operation of the individual modules is described in a separate section. All operations of the A3 amplifier are described in one section. W6 descriptions also apply to the W2 except where otherwise noted.

Output Termination

Output signals must be properly terminated to minimize signal reflection or power loss due to an impedance mismatch. Proper termination is also required for an accurate amplitude level at the main output connector. Use 50 Ω cables and terminate the main and SYNC cables with terminating resistors. Use a 50 Ω termination at the far end of the cable if needed to improve impedance matching.

The accuracy of the amplitude setting depends upon the load impedance. The amplitude of the output signal will not match the amplitude setting for loads that differ from 50 Ω. To correct for this error, program the actual load impedance and let the 3172 automatically correct its signal amplitude. Use the following command:

outp:load <value>

where <value> is the load impedance in ohms.

Input/Output Protection

The 3172 provides protection for its internal circuitry at the input and output connectors. Appendix A specifies the level of protection for each input or output connector.

Power On/Reset Defaults

At power-up or as a result of a software reset, the 3172 changes all settings to their default values. Chapter 4 lists all settings and their default values, as well as their maximum and minimum allowable values.

It is good practice to reset the instrument between sequence events to make sure that parameters that were programmed for previous tests will not accidently interfere with future tests. Use the following common command to reset the instrument to its default state:

**rst*

Turning the Output On

For safety, the “OUTPUT” connector of the 3172 defaults to the “Off” state when the unit is first powered up or receives a software reset. To avoid the production of random signals at the output connector, keep the output in the “Off” state while setting up the instrument.

The output signal is connected to the “OUTPUT” connector through a mechanical relay. When writing software to control the 3172, provide a 2 ms delay to make sure that the signal output is stable before you proceed with the next step of the test. Use the following commands to turn the output on and off.

outp 1	Turns the output on
outp 0	Turns the output off

The W6 defaults to a 1 MHz, 5 Vp-p sine wave when you apply power or reset the unit. If you turn the output on before changing any settings, the output defaults to a sine wave signal.

Turning the SYNC Output On

For safety, the “SYNC OUT” connector of the 3172 defaults to the “Off” state when the unit is first powered up or receives a software reset. To avoid the production of random signals at the output connector, keep the SYNC output in the “Off” state while setting up the instrument.

The SYNC output signal is hard-wired to the output connector and, therefore, the output impedance remains low regardless of whether the output is on or off. Use the following command to turn the SYNC output on and off:

Outp:sync 1	Turns the SYNC output on
Outp:sync 0	Turns the SYNC output off

When you apply power or reset the 3172, the SYNC output defaults to a single pulse that has a fixed width of four sample clock periods.

Selecting the SYNC Source (W6 Modules Only)

The main purpose of the SYNC output is to produce a pulse that other devices may synchronize with. The width of the “SYNC” signal might be too narrow for slower devices. The pulse width, and several other parameters, may be programmed using the following commands:

outp:sync:sour bit Provides a signal that is synchronized with an external trigger event when the W6 is placed in one of its interrupted modes. Generates a narrow sync signal every time the segment is generated. The sync position along the waveform can be programmed using the OUTP:SYNC:POS command. OUTP:SYNC:POS is used to set both the TRIGger point and the SYNC point. The BIT signal is recommended for use in continuous mode.

outp:sync:sour lcom Generates a sync signal in SEQuence mode only once when the selected segment appears for the first time in the sequence. The identity of the segment can be programmed using the TRAC:SEL command. The sync position along the selected waveform can be programmed using the OUTP:SYNC:POS command. The LCOM signal is recommended for use in Sequenced mode.

outp:sync:sour ssyn Generates a sync signal at intervals that are synchronized with the internal clock generator. This option is useful to minimize jitter when using an oscilloscope. The SSYNc signal is recommended for use in Triggered mode.

outp:sync:sour puls Generates a pulse each time a segment waveform is generated. The pulse width is specified in points using the *OUTP:SYNC:WIDTh* command. This command is a useful alternative to the BIT sync source, especially when the bit pulse is too narrow. It is also helpful when using the W6 to emulate the sync pulse of another instrument that it is replacing in a test system.

outp:sync:sour zero Generates a SYNC signal which remains low when the main output level is below 0 V and shifts to high when the output signal becomes greater than 0 V.

Changing the SYNC Position and Width (W6 Modules Only)

The “SYNC” signal must be wide enough for other devices to detect, but not so wide that it adversely affects timing in faster applications. To accommodate a wide variety of situations, the SYNC pulse width is programmable.

The programmed start point for the SYNC signal is in effect for all of the SYNC types except ZERO. The SYNC width parameter is in effect only when the pulse type is selected. Use the following procedure to program SYNC position and width.

outp:sync:sour puls	Selects pulse as the sync source. In this mode, you may also program the position of the sync pulse.
outp:sync:pos <value>	Sets the position of the sync pulse relative to the active segment. The position is programmed in units of waveform points.
outp:sync:wid <value>	Sets the width of the sync pulse. The width is in units of waveform points. Note that the position (in points) plus the width (in points) shall not exceed the number of points in the segment.

Example: Generating a Simple Waveform

This example assumes that you have set up the chassis and run the Resource Manager.

Connect two cables, one from the OUTPUT connector and the other from the “SYNC” connector, to an oscilloscope. Set up the oscilloscope to trigger from the SYNC signal. Use the following sequence of commands to set up the 3172:

**rst*	Restore factory defaults (Table 5-1 provides a complete listing of defaults)
outp 1	Enable output relay to turn output on
volt <value>	Set up the amplitude level
volt:offs <value>	Set up the offset level

Set up your oscilloscope to observe that the 3172 generates a sine waveform with the following properties:

On W6 modules:

Frequency: 1 MHz
Offset: 0 V
Amplitude: 10 V

On P2 modules:

Waveform: Pulse

Period: 1 ms
Offset: 0 V
Amplitude: 10 V

The output of the 3172 is calibrated for signals applied to a 50 Ω load. If your amplitude is twice as higher than expected, then the 3172 output may not be properly terminated. In this case, either add a 50 Ω termination to the cabling or change your oscilloscope settings so that its input uses the built-in 50 Ω input termination. Also, note the interaction between amplitude and the offset as described below.

Programming Amplitude and Offset

The 3172 amplitude and offset can be programmed within an amplitude window of 5 mVp-p up to 22 Vp-p (20 Vp-p for POS or NEG ranges). This window can be shifted and used in three amplitude ranges:

- Positive, where signals can be programmed from 0 V to 20 V;
- Negative, where signals can be programmed from 0 V to -16V;
- Symmetrical, for signal span from -11 V to 11 V

Regardless of the amplitude range that you choose, values may not exceed the window maximum. Always watch your combined amplitude and offset settings because if you exceed the window maximum, the combined values will cause a settings conflict.

Amplitude and offset may be programmed freely as long as the following relationships are observed:

POS or NEG range:

$$\frac{Amplitude}{2} + |Offset| \le 20Vp-p$$

SYMM range:

$$\frac{Amplitude}{2} + |Offset| \le 22Vp-p$$

Amplitude-offset combinations outside the above limits will generate "settings conflict" errors.

Use the following sequence of commands to select an amplitude range and then program the amplitude and offset values:

**rst*	Restore factory defaults (Table 5-1 provides a complete listing of defaults)
outp 1	Enable output relay to turn output on
volt:rang <value>	Set up an amplitude range: *pos, neg or symm*
volt <value>	Set up the amplitude level

volt:offs <value>	Set up the offset level

Note that when an amplitude range is changed, the peak-to-peak amplitude and offset values automatically default to the factory preset values.

Programming the Output Impedance

The 3172 has three output impedance options to match industry standard load impedances: <2 Ω, 50 Ω and 93 Ω. The output is calibrated on a matching load impedance and, therefore, the appropriate output impedance for the actual load should be selected for specification compliance. Use the following sequence of commands to select an output voltage window, output load impedance and then program the amplitude and offset values:

**rst*	Restore factory defaults (Table 5-1 provides a complete listing of defaults)
outp 1	Enable output relay to turn output on
outp:imp <value>	Set up the output impedance: *0, 50 or 93*
volt:rang <value>	Set up an amplitude range: *pos, neg or symm*
volt <value>	Set up the amplitude level
volt:offs <value>	Set up the offset level

If your load impedance is not matched to the output impedance, the amplitude reading at the load will differ from the programmed value. The 50 Ω setting is the default setting and is recommended for use in most applications that require a good pulse response and minimal aberrations.

Selecting an Output Function

The 3172-W6 has four basic output function types. Use the following commands to select the waveform type:

func:mode fix	The 3172-W6 outputs the standard waveform currently selected by the FUNC:SHAP command.
func:mode user	The 3172-W6 outputs the arbitrary waveform currently selected by the TRAC:SEL command.
func:mode seq	Selects the Sequenced Waveform function. Waveform segments must be downloaded to the instrument before it can sequenced generate waveforms. In addition, the SEQ:DEF command must be used to define the sequence.

When programming a waveform function, you must consider the run mode, since combinations of waveform types that are legal in one mode may not be legal in a different mode. For example, arbitrary and sequenced waveforms that do not conflict in Continuous mode may conflict in Burst mode.

Selecting a Run Mode

The 3172 offers five run modes:

Continuous
Triggered
Re-triggered
Gated
Burst

In Continuous mode, the 3172 repeats the selected waveform continuously. The other four modes are known as *interrupted*, that is, the output does not generate waveforms until an event initiates a single- or multiple-output cycle.

Interrupted modes require a single trigger or multiple triggers to initiate output cycles. Trigger events come from various inputs, such as VXIbus backplane trigger lines or the front-panel connector. Information on selecting a run mode and trigger source is given in the following paragraphs.

Use the following commands to select run modes:

init:cont on	Selects Continuous mode. This is the default state of the instrument. Use this command to return to Continuous run mode from any other interrupted mode.
init:cont off	De-selects Continuous mode. If you did not previously select a specific interrupted mode, the generator will automatically be placed in Triggered mode.

Continuous Run Mode

Upon power-up, the 3172 defaults to Continuous mode and automatically starts generating waveforms which appear at the output connector as the output relay is turned on. Use “init:cont on” to return to Continuous mode from any interrupted mode.

Triggered Run Mode

In Triggered mode, the output remains at a specific DC level until a trigger event initiates a single output cycle (see “Selecting the Trigger Source” for details about trigger parameters).

Each time a transition occurs at the trigger input, the 3172 generates one complete output cycle. At the end of the waveform cycle, the output assumes a DC level that is equal to the amplitude of the first point of the waveform.

If you have not invoked any interrupted modes since applying power to the 3172, you may use the following command to turn off Continuous mode and default to Triggered mode:

init:cont off	Turns off Continuous mode, changing to an interrupted mode. If you have not selected a specific interrupted run mode since power-up, the generator will default to Triggered mode.

If you have used other interrupted modes since power-up, then send the following command sequence to guarantee a change to Triggered mode:

init:cont off	Turns Continuous mode off in case this was the most recent run mode.
trig:retr off	Turns Retriggered mode off in case this was the most recent interrupted run mode.
trig:gated off	Turns Gated mode off in case this was the most recent interrupted run mode.
trig:burs off	Turns Burst mode off in case this was the most recent interrupted run mode.

Re-Triggered Run Mode

Re-triggered mode is a special case of Triggered mode where, after a valid trigger event, the generator automatically self-issues triggers separated by a programmable delay. The delay interval is measured from the end point of the waveform to the start point of the next waveform cycle. Use the following commands to place the instrument in re-triggered mode and to program the delay between events:

init:cont off	Turns Continuous mode off, changing to an interrupted mode. If you have not selected a specific interrupted run mode since power-up, the generator will automatically be placed in Triggered mode.
trig:retr 1	Turns on Re-triggered mode.
trig:retr:del <value>	Sets the re-trigger delay time. The re-trigger delay is measured from the last point of the waveform cycle to the first point of the next waveform cycle.
<trigger event>	Starts the re-trigger generator. You may select the source of trigger events from a number of inputs, such as software trigger, front-panel input, or VXIbus backplane trigger lines.

The waveform will continue to re-trigger unless you change the run mode or turn off power. Use the following commands to restore continuous run mode:

trig:retr 0	This turns off Re-triggered mode. The generator will revert to the Triggered run mode.
init:cont on	This removes the 3172-W6 from interrupted run mode and reverts to Continuous mode.

Gated Run Mode

In Gated mode, the output remains at a specific DC level until a valid event opens the gate. Only triggers from hardware sources can open and close the gate. Use VXIbus backplane trigger lines or the front-panel trigger input as the gating control.

You may set the edge sensitivity of the trigger signal for either the rising or falling edge of the signal. At the end of the last output cycle, the output assumes a DC level equal to the amplitude of the last point of the waveform.

There are two selectable conditions for opening the gate:

Two transitions in the same direction toggle the gate on and off.

The gate remains closed as long as the trigger signal is below the trigger level setting, and opens when the trigger signal exceeds the trigger level setting.

Use the following commands to turn the gate function on and to select the condition that will open the gate:

init:cont off	Selects the interrupted run mode. If you did not select a specific interrupted run mode since power-up, the generator will automatically be placed in Triggered mode.
trig:gate 1	Turns the Gated mode on.
trig:gate lev	This option makes the gate level-sensitive. The gate opens when the gating signal amplitude exceeds the value of the programmed trigger level. This is the default for Gated mode.
trig:gate tran	This option makes the input transition-sensitive. The gate opens on the first transition, and closes on the next transition.
<gate event>	This event controls Gated mode. You may select the source of the gate events from a number of inputs, including the front-panel or VXIbus backplane trigger line.

The gating sequence continues unless you change the run mode or turn off power. Use the following commands to restore Continuous mode:

trig:gate 0	Turns off the Gated mode. The generator reverts to Triggered mode.
init:cont on	Places the 3172 into Continuous mode.

Burst Run Mode

Burst mode is similar to Triggered mode except that only one trigger signal is needed to generate a counted number of output cycles. In Burst mode, the output remains at a specific DC level until a valid trigger event initiates a burst of output waveforms. Any trigger source can initiate a burst. If a hardware trigger source is selected, the edge sensitivity can be programmed for either the rising or falling edge of the input signal.

Each time a transition at the trigger input occurs, the 3172 generates a counted burst of output waveforms. At the end of the burst, the output assumes a DC level equal to the amplitude of the first point of the waveform. The burst counter is programmable from 1 to 1 M counts.

The 3172 can also operate in conjunction with Re-triggered mode, creating a continuous sequence of delayed burst cycles. Use the following commands to place the instrument in Burst mode and to program the burst counter:

init:cont off	Selects the interrupted run mode. If you have not selected a specific interrupted run

mode since power-up, the generator will automatically be placed in triggered run mode.

trig:burs 1	Turns the Burst mode on.
trig:burs:coun <value>	Sets the burst counter. After a legal trigger event, the instrument will generate the counted number of waveforms, and then resume idling at a DC level.
<burst event>	This starts the burst generator. You may select the source of the burst event from a number of inputs including software trigger, front-panel input, or VXIbus backplane trigger line.

The counted burst sequence will continue as long as legal trigger events are present at the trigger input. Use the following commands to restore Continuous mode:

trig:burs 0	This turns off Burst mode. The generator reverts to Triggered run mode.
init:cont on	This removes the 3172-W6 from an interrupted run mode and reverts to Continuous run mode.

Selecting the Trigger Source

Interrupted run modes require trigger signals, of which there are three types:

External trigger. For synchronizing with external events, connect the external trigger signal to one of the trigger inputs.

Internal trigger generator. For applications not requiring synchronization to external signals, the 3172 has an internal trigger generator with a free-running clock. This clock is asynchronous to the sample clock generator.

Software trigger. Your software may generate an interrupt condition by executing a trigger command.

You may select (arm) only one trigger source at a time. The 3172 responds only to the selected trigger source, and ignores other sources. Use one of the following commands to select a trigger source:

trig:sour ext	This selects the front-panel TRIG IN connector as the active source for trigger events.
trig:sour int	This selects the internal trigger generator as the active source for trigger events.
trig:tim <value>	This sets the period of the built-in trigger generator. Unlike the Re-trigger run mode, the internal trigger period defines actual trigger events. Therefore, the period of the

internal trigger generator must be larger than the period of the waveform.

trig:sour ttlt<n> This selects and activates one or more of the VXIbus backplane triggers (TTLTrg0 through TTLTrg7) as the active source for trigger events. If more than one input is activated, the instrument will accept trigger events from all active trigger lines. Note that if one of the lines is designated as an output, it cannot be used as an active source at the same time.

trig:sour bus This selects the software trigger as the active source for trigger events.

Selecting the Trigger Level

The Trigger Level command sets the threshold level for the trigger input connector only. The trigger level is adjustable from -10 V to +10 V using the following command:

Trig:lev <value> This programs the trigger level threshold for signals that are applied to the front-panel TRIG IN connector.

The default value is 1.6 V which is appropriate for TTL signals.

Selecting the Trigger Slope

The Trigger Slope command selects between positive- and negative-edge triggering. The inputs that will be affected by this command are: Front-panel TRIG IN connector, TTLTrg lines 0 through 7, and ECLTrg line 0. Use the following command to select slope sensitivity for trigger events:

trig:slop pos This sets the 3172 to respond to positive going transitions only. Positive transitions must cross the trigger level threshold to trigger a response.

trig:slop neg This sets the 3172 to respond to negative going transitions only. Negative transitions must cross the trigger level threshold to trigger a response.

Using Trigger Delay

The trigger delay value designates the time that will elapse from a trigger event to the start of the waveform at the output connector. The trigger delay adds to the system delay time (see the definition of System Delay in Appendix A). Therefore, when delaying the trigger, always consider the added factor of the system delay.

Use the following command to turn on trigger delay and to program the delay time value:

trig:del:stat 1	This command turns on the trigger delay function. When turned on, the minimum delay time is 100 ns plus system delay.
trig:del:stat 0	This command turns off the trigger delay function. When turned off, the minimum delay time is equal to the system delay.
trig:del:tim <value>	This sets the delay in units of time (an alternative to the legacy compatible method above which uses units of sample clock points). You may set the delay time in the range of 200 ns to 20 seconds in increments of 20 ns.

Activating the Backplane TTLTrg Lines

The 3172 is programmable to drive or receive triggers on the VXIbus backplane trigger lines (TTLTrg0 through TTLTrg7). It is important to manage the trigger lines so that only one VXI instrument drives any given line.

The signals through the TTLTrg lines are always TTL.

Use the following commands to program the backplane trigger lines.

outp:ttlt<n> 1	This command defines a specific TTLTrg line as an output. The SYNC signal is then applied to the active TTLTrg output and the front-panel SYNC OUT connector. The argument <n> designates the required TTLTrg line, and can take values from 0 through 7.
outp:ttlt<n> 0	This command removes the output definition from a specific TTLTrg line and permits re-definition of this line as an input.

Controlling W6 Function and Parameters

The descriptions up to this point have been mostly common between the W6 and P2. Description from this point on pertains to the W6 module only. Later in this chapter, there will begin a section which describes the operation of the P2 module.

Selecting an Output Function

The W6 has six basic output functions; these are: Standard Waveforms, Arbitrary Waveforms, Sequenced Waveforms, Modulated Waveforms and Half-Cycle Waveforms. Use the following commands to select the output function:

func:mode fix	The W6 outputs the standard waveform

	currently selected by the FUNC:SHAP command. Standard functions are re-computed every time a new function or is selected or parameter is modified.
func:mode user	The W6 outputs the arbitrary waveform currently selected by the TRAC:SEL command.
func:mode seq	Selects the sequenced waveform function. Waveform segments must be downloaded to the instrument before it can sequenced generate waveforms. In addition, the SEQ:DEF command must be used to define the sequence.
func:mode mod	Selects the modulated waveform function. Modulation schemes are generated internally by a special DDS circuit and are routed through a by-pass leg to the output connector. The MOD:TYP command is used for selecting the required modulation scheme.
func:mode half	Selects the half cycle waveform function. Similar to the standard waveform function, half cycle functions are recomputed every time this function is selected or a parameter has changed.

When programming a waveform function, you must consider the run mode, since combinations of waveform types that are legal in one mode may not be legal in a different mode. For example, arbitrary and sequenced waveforms that do not conflict in Continuous mode may conflict in Burst mode.

Example: Generating Standard Waveforms

Previous paragraphs provided sinusoidal waveform examples, showing how to set amplitude and offset. This section expands on that capability, covering all nine standard waveforms in the 3172-W6 internal library: sine, triangle, square, pulse, ramp, Gaussian pulses, exponential pulses, DC, and noise.

The following command sequence example demonstrates how to select a standard ramp waveform with an amplitude of 1Vp-p, an offset of zero, a rise time of 10% of the ramp period, and a fall time of 10% of the ramp period:

**rst*	Restores factory defaults.
outp 1	Activates the hardware connection to the front panel OUTPUT connector and turns the output on.
func:mode fix	Selects the built-in library of standard

waveforms.

func:shap ramp	Selects the ramp from the built-in library as the active waveform.
ramp:tran 10	Programs the ramp leading edge rise time in units of percent (referenced to the ramp period).
ramp:tran:tra 10	Programs the ramp trailing edge fall time in units of percent (referenced to the ramp period).
volt 1	Sets the amplitude level to 1Vp-p.
volt:offs 0	Sets the offset level to zero.

Similar sequences could select different standard waveforms and program their parameters. See Chapter 5 for a programming reference to the complete range of standard waveforms.

Generating Standard Waveforms

Each standard waveform is built into the 3172-W6 in a lookup table or equation. Ten standard function shapes are available:

Sine
Triangle
Square
Pulse
Ramp
Sinc
Exponential Decaying Pulse
Gaussian Pulse
Noise
DC

Every time you select a standard function, the 3172-W6 retrieves the data points from a lookup table or calculates them from equations, and then places them into waveform memory.

Use the following commands to select one of the standard waveform shapes:

func:shap sin	Selects the sine waveform
func:shap tri	Selects the triangle waveform
func:shap squ	Selects the square waveform
func:shap puls	Selects the pulse waveform
func:shap ramp	Selects the ramp waveform
func:shap sinc	Selects the sinc waveform
func:shap gaus	Selects the Gaussian waveform
func:shap exp	Selects the exponential waveform
func:shap nois	Selects the noise waveform

func:shap dc Selects the DC waveform

After you select a waveform shape, you may specify the waveform parameters. Chapter 5 explains the commands available for specifying parameters of standard waveforms.

Note

The number of points used for defining a standard waveform depends upon the programmed frequency. Therefore, some parameter changes may not have any effect on the waveform because the number of points do not provide enough resolution to show the difference.

The number of waveform points varies, depending upon the output frequency. The reason for this variation is that even standard waveforms are in a manner similar to that of arbitrary waveforms, except that the 3172-W6 stores standard waveforms in a permanent internal library for immediate use. At low frequencies, the number of points for each standard waveform is 1,000. Therefore, waveform modifications are possible in increments of 1/1,000 of the total waveform. For example, if you want to modify the duty cycle of a square waveform, the resolution for the duty cycle is 0.1%.

At higher frequencies, the number of points used for generating waveforms decreases according to the following relationship:

Output Frequency = Sample Clock Frequency / Waveform Points

Since the maximum sample clock frequency is 200 MS/s, the only way to increase frequency is by reducing the number of waveform points (when the standard waveform frequency is > 200 kHz). The reduced number of points available at higher frequencies decreases timing resolution for the standard waveforms. For example, for a 20 MHz square wave, only ten points per period are available. Therefore, duty cycle resolution is decreased to 10% increments.

Standard Waveform Parameters

The built-in library of standard waveforms provides basic waveform shapes. First select the basic shape, and then specify the waveform parameters to create the finished waveform to fit your requirements. For each standard waveform shape, you may adjust the frequency, amplitude, and offset. Some wave shapes have additional parameters available. For example, you may set the starting phase for a sine wave, or the rise and fall time for a ramp waveform.

The following example demonstrates how to select a standard pulse waveform and set its parameters:

**rst*	Restores factory defaults.
func:mode fix	Sets the output function to the standard built-in library of waveforms. This is also the default function mode so if you are using the reset command, there is no need to re-select this function.
func:shap puls	Selects the pulse waveform
puls:del 0	Sets the delay value to 0 s
puls:wid 20	Sets the pulse width to 10%, expressed as a percentage of the waveform period.
puls:tran 3	Sets the rise time (leading edge) to 3%, expressed as a percentage of the waveform period.
puls:tran:tra 5	Sets the fall time (trailing edge) to 5%, expressed as a percentage of the waveform period.
freq 5e3	Sets the frequency of the pulse waveform to 5 kHz (5e3 Hz).
outp 1	Turns the output on.

You may use similar command sequences to program other waveforms and their parameters. Table 5-1 lists the complete set of commands for setting waveform parameters.

Using the Apply Command

The Apply command is a shortcut for setting up standard waveforms and their parameters without having to program each parameter individually. This also selects the waveform as the active signal at the output connector. It does not, however, eliminate the need for turning on the output. You may use the Apply command on a waveform from the built-in library or on waveform segments that are pre-loaded with arbitrary waveforms. Sequenced and modulated waveforms are not supported by the Apply command.

The following example uses the Apply command to specify a square wave at the end of the programming sequence:

**rst*	Restores factory defaults.
appl:squ 10.7e6,2,1,30	Selects the standard square wave as the active function, and simultaneously sets the frequency to 10.7 MHz (10.7e6), amplitude to 2 V, offset to 1 V, and duty cycle to 30%.
outp 1	Turns on the output.

The above is an example of a full utilization of the Apply command, including the frequency, amplitude, offset, and duty cycle parameters for a standard square wave. You may use the Apply command in a similar manner for other standard or arbitrary waveforms.

You may use the Apply command on a partial set of the available parameters of a waveform. To leave out a parameter, place no characters between the corresponding comma separators. For example, you may specify the frequency and duty cycle of a square wave while leaving the amplitude and offset at the default values, as follows:

**rst*	Restores factory defaults.
appl:squ 12.7e6,,,45	Selects the standard square wave as the active function, and simultaneously sets the frequency to 12.7 MHz, leaves the amplitude at the default value, leaves offset at the default value, and sets the duty cycle to 45%.
outp 1	Turns on the output.

In a similar manner, you may use the Apply command with other standard waveforms to set some parameters while leaving others at their default values.

Generating Arbitrary Waveforms

Before the 3172-W6 can generate arbitrary waveforms, you must first download them to its waveform memory. This section describes the arbitrary waveform function and explains how to download waveforms.

What are Arbitrary Waveforms?

Arbitrary waveforms are generated from digital data points which are stored in memory. Each data point (waveform sample) has a vertical resolution of 16 bits (65,536 levels). Another way to express this is that each sample has an amplitude resolution of one part in 65,536. For legacy emulation, 12 bit waveform data is converted into 16 bit data with a four position shift.

The standard 3172-W6 has a waveform memory capacity of 1 M points. Each point has a unique address. The address of the first point is zero, and the address of the last point depends upon the waveform memory size. If a waveform does not require the entire waveform memory, then you may divide the memory into smaller segments, each of which may store a separate waveform.

When the instrument is set to output arbitrary waveforms, the clock samples the data points one at a time, starting with address 0 and continuing to the last data point of the waveform. The rate at which each sample is retrieved is defined as the sample clock rate. The 3172-W6 provides programmable sample clock rates from 100 mS/s to 200 MS/s.

Unlike waveforms contained in the built-in library, arbitrary waveforms must first be downloaded into waveform memory. One of the easiest ways to calculate the waveform samples is to use ArbConnection. It provides an on-screen editor, called Wave Composer, for creating and editing waveforms. Figure 3 -1 shows a complex waveform from the Wave Composer editor. Chapter 4 provides instructions for using ArbConnection and its Wave Composer editor.

Managing Arbitrary Waveform Memory

You may divide the 3172-W6 waveform memory into segments, and use each segment to contain a separate waveform. This is useful in applications that require multiple waveforms and can benefit from changing quickly from one waveform to another.

The memory can be partitioned into as many as 16k segments (with up to 16,384 different waveforms), but the higher the number of segments, the smaller the number of sample points that are available to each.

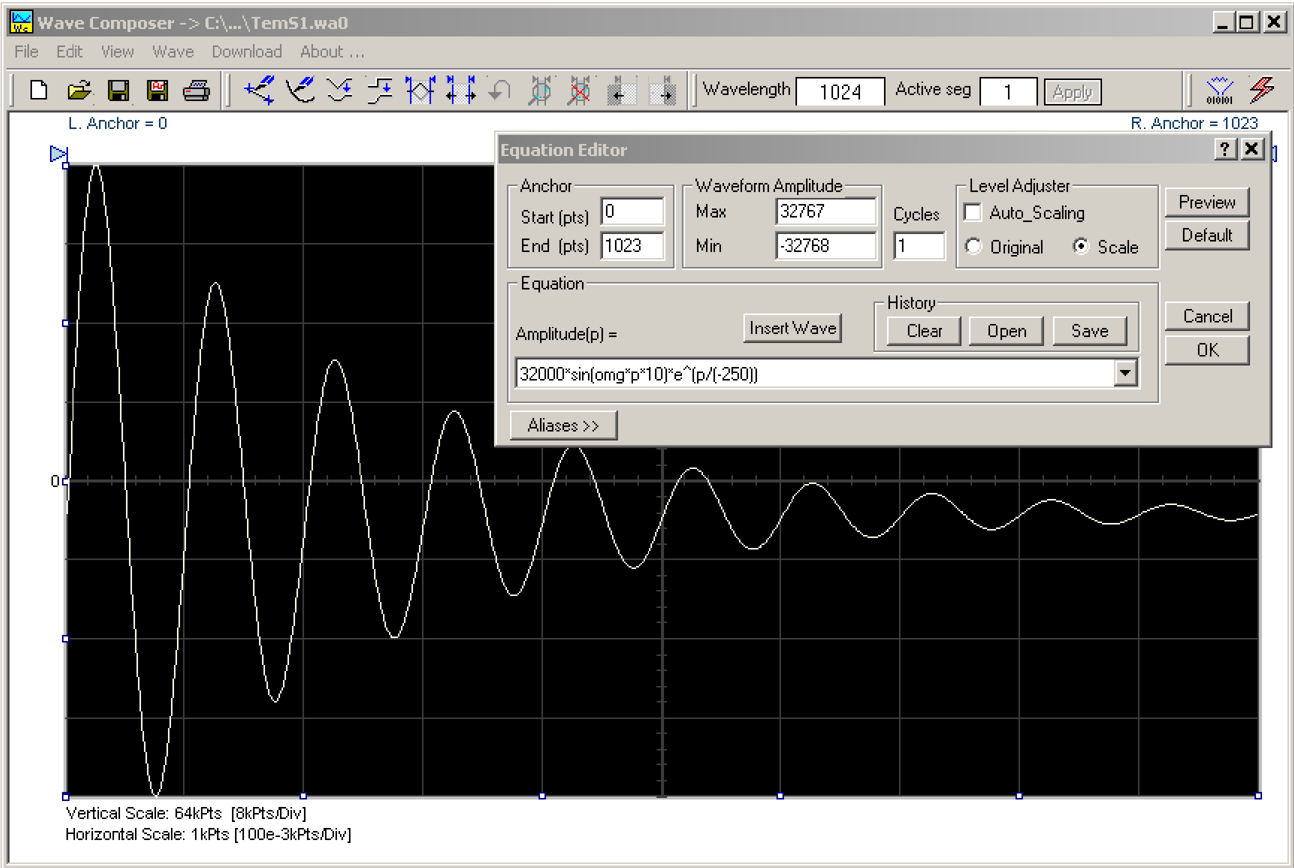

Figure 3-1, ArbConnection Example of a Complex Waveform

Memory Management Commands

Creating Memory Segments

Segments are defined using the following command:

trac:def 1,2000	Defines segment #1 as having 2,000 sample points. Any waveform downloaded to this segment must have exactly 2,000 data points.

This command has two variables: segment number and segment size. Note that numbers, not names, are assigned to segments. Segment numbers range from 1 through 16,384. You may define the segments in any order. For example, you may define segment #3, then segment #1, and then later define segment #2. You may not change the size of a segment once you have defined it.

You cannot query the segment definition parameters, so make sure you keep track of them if you intend to partition the memory into many segments.

You may use the above command to create as many segments as required. However, if you have many segments, it is more efficient to combine all segments into a single waveform, and then create a memory partition table for the individual waveform segments. To do this, use the following command:

segm <array>	Downloads the entire memory partition table to the instrument in one operation.

Chapter 5 provides details on the use of this command.

Deleting Memory Segments

To delete a waveform memory segment, use the following command:

trac:del <value>	Deletes a segment (specified by <value>) from the available segment list but does not erase the contents of the segment.

Note that if you delete a segment, the memory portion that belonged to this segment is no longer accessible. The next segment defined is placed at the end of the partition table. If you delete the last segment that you defined, then the next downloaded data will overwrite the memory of the deleted segment. If you delete segments often, large portions of the memory will become inaccessible. Therefore, it is suggested that you periodically clear the entire memory and reload the waveforms that you intend to use. To delete the entire memory partition table use the following command:

trac:del:all	Removes the entire partition table, allowing definition of a new segment table.

Tip

The *trac:del:all* command deletes the partition table but leaves the data in the arbitrary memory intact. Therefore, if you made a mistake and want to restore the segments, just re-load the partition table. The waveform memory is overwritten. Every time you download new waveform data, the waveform memory data for that segment is overwritten.

Loading Arbitrary Waveforms

The easiest way to download waveforms to the 3172-W6 is with ArbConnection. Using this application, you may define, create, and download memory segments to the 3172-W6.

For maximum flexibility, you may download waveforms to the 3172-W6 from your own program. The following example shows how to clear the partition table and set up three memory segments of 4,000, 1,000, and 64 sample points:

First, clear the entire memory partition table to eliminate any fragmented segments. Use the delete command as follows:

trac:del:all	Removes the entire partition table and

allows you to define new segments.

Next, define the waveform memory segments by specifying each segment number and its length:

trac:def 1,4000	Defines the length of segment #1 to be 4,000 sample points. Waveforms downloaded to this segment must have exactly 4,000 sample points.
trac:def 2,1000	Defines the length of segment #2 to be 1,000. Waveforms downloaded to this segment must have exactly 1,000 sample points.
trac:def 3,64	Defines the length of segment #3 to be 64. Waveforms downloaded to this segment must have exactly 1,000 sample points.

This completes the setup for the three memory segments.

Alternatively, you may use the following command to create the entire partition table at once:

segm <array>	Downloads the entire memory partition table to the instrument in one operation.

Once you have defined the waveform memory segments, the next step is to specify the active segment. This sets up the following conditions:

1) The next time you download data to the 3172-W6, it will go to the active segment (the 3172-W6 will accept downloaded data only if a segment is designated as active).
2) The waveform contained in the active segment will appear at the output the next time you turn on the output.
3) The SYNC output is associated with the active segment. This is not important in arbitrary mode, but in sequenced mode, the segments may be arranged in any order. Therefore, the location of the sync signal is important because it may appear at the middle of the sequence and not with the first segment.

Use the following command to select an active segment:

trac:sel <n>	Selects the active segment <n>. Waveform data is downloaded only to this active segment. If you plan to partition the entire table with the “segm <array>” command, select segment #1 as the active segment.

The next step transfers waveform data to the active segment. Use the following command:

trac# <data_array>	Downloads waveform sample data to the active segment. If you condensed all waveforms to a single waveform,

then you may use the “segm <array>” command to partition the memory into segments in one operation. See Chapter 5 for information on preparing and downloading waveform sample data.

Changing the Sample Clock Frequency

Users should be careful not to confuse waveform frequency with sample clock frequency. For the 3172-W6, the term “waveform frequency” is valid for standard waveforms only, and controls the waveform frequency at the output connector. Waveform frequency is measured in units of Hertz (Hz).

On the other hand, the term “sample clock frequency” is associated with arbitrary and sequenced waveforms only, and defines the frequency at which the clock generator accesses the waveform sample points. Sample clock frequency is measured in units of samples per second (S/s).

The following equation computes the frequency of an arbitrary waveform at the output connector:

Frequency = (Sample Clock Freq.) / (Number of Data Points)

For example, using a sample clock frequency of 80 MS/s with a 1,000-point waveform will generate an 80 kHz waveform at the output connector.

The following command sets the sample clock frequency for arbitrary and sequenced waveforms:

freq:rast <value>	Set sample clock frequency in units of samples per second

Sampling clock frequency can be in the range of 100 mS/s to 200 MS/s.

The number of points in the waveform must be an integer multiple of four. For example, you may use a waveform length of 25,804 throughout the entire range, but if you increase the number of points by two, then the 3172-W6 will generate an error.

Using the External Sample Clock Input

The internal sample clock generator has a wide dynamic range that allows the creation of an infinite number of waveforms and frequencies. With its top frequency reaching frequencies close to 200 MHz, it must use dividers to create lower frequencies. Such dividers can increase phase noise and jitter. Some applications require better stability and phase noise, making a single-tone sample clock source the most desired source.

The 3172-W6 does not have a single-tone sample clock source, but it provides a front-panel input, that can accept a clock from an external source. When this input is in use, the internal clock generator is disabled, and the 3172-W6 waveforms are clocked at a rate defined by the external signal. Using an external clock source can improve phase noise and jitter to approximately 20 dB/Hz at 10 kHz offset from the carrier.

Apply the external sample clock signal to the front panel SCLK IN connector. Make sure your signal level is within the levels specified in Appendix A. The following commands select the source of the sample clock input:

freq:rast:sour int This is the default selection, where the 3172-W6 self-generates its sample clock signal. All other inputs are disabled.

freq:rast:sour ext This selects the front panel SCLK IN connector as the source of the sample clock signal. Observe the signal range and levels as specified in Appendix A.

freq:rast:sour eclt0 This selects the backplane ECLTrg0 line as the sample clock source. This is a special mode that allows synchronization between adjacent 3172-W6 modules. Note that the VXI specifications limit the ECLTrg0 frequency to 62.5 MHz. Backplane synchronization is covered in a separate section.

Generating Sequenced Waveforms

Sequences are comprised of waveform segments that reside in the waveform memory. The sequence generator lets you link and loop segments in a user-defined order. To avoid unexpected results, it is essential that waveform segments are pre-loaded into waveform memory before a sequence table is used.

To create a waveform sequence, you will create a sequence table that provides instructions to the sequencer for assembling the waveform from the segments in the waveform memory. Figure 3-2 shows an example of a sequence table created using ArbConnection. The sequence table has five fields for each step:

Link – This defines the step number. The sequence will advance through the links in the same order in which they are entered. There are no pauses or transitions between links.

Seg – This defines the waveform segment number that will be linked to form the next part of the sequence. The order of waveform segments in waveform memory is irrelevant. You may link them in any order.

Loops – This defines the number of times the segment will repeat itself before advancing to the next link, or step. The number of loops may range from 1 through 1,048,576.

Adv – Defines the advance bit, which tells the generator whether to move to the next link immediately or to hold and wait for a trigger event before moving to the next link. The various advance modes are discussed in more detail in another section.

Sync – This specifies the link on which you want to place a synchronization bit. The SYNC output must be switched to the Bit source option to enable the use of this feature. The Normal SYNC output is LCOM, where the output goes high at the beginning of the sequence and returns to low at the end of the sequence.

The following paragraphs describe the commands that create, delete, update, and modify sequence tables.

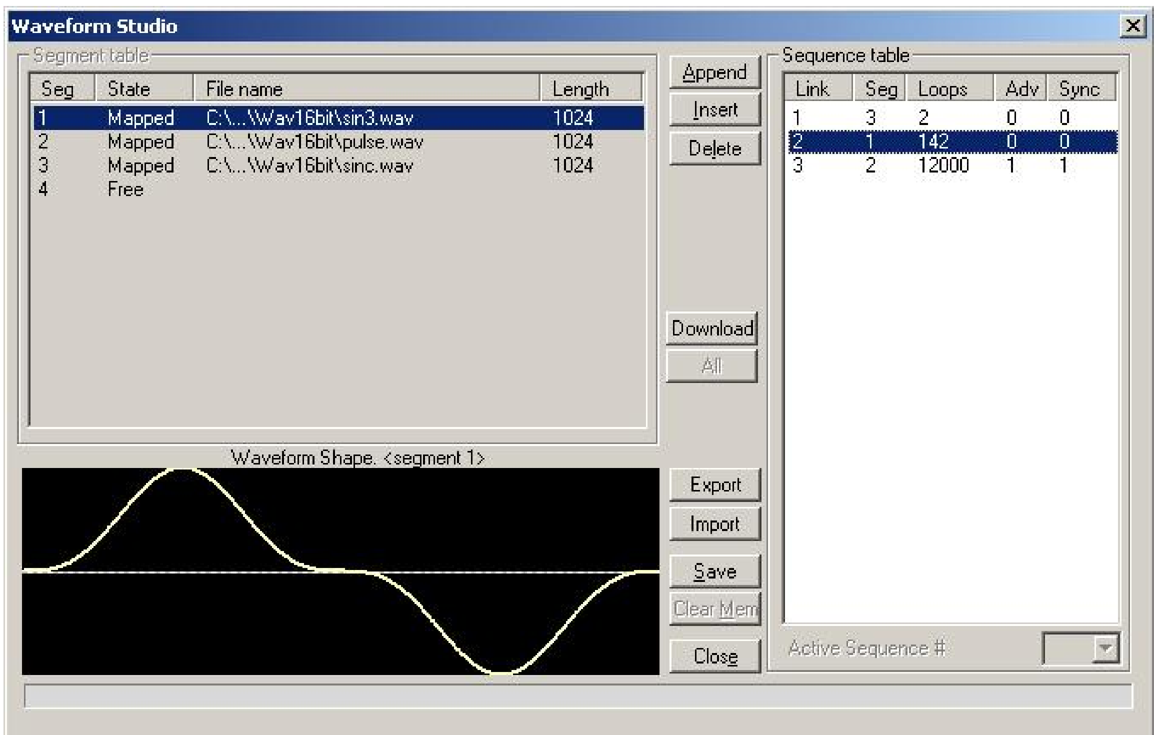

Figure 3-2, Sequence Table Created in ArbConnection

Sequence Commands

The following is an overview of how to define and program a sequence of arbitrary waveforms.

A sequence is made of a series of links. A link can stand on its own or link to another step. It is possible to have only one link in a sequence, but the output will be a continuous waveform. If only one link is specified and the 3172-W6 is placed in Triggered advance mode, then the output will behave as it would in Burst mode, where the repeat number replaces the burst count parameter.

The easiest way to create a sequence table is with ArbConnection. Using this application you can define, create, and download waveform segments to the waveform memory without using low level commands. You may also use Waveform Studio to develop sequences.

Use the following commands to write sequence tables:

seq:sel <1..10>	The 3172-W6 can store ten separate sequences, and this command selects the active sequence. Once selected, the active sequence is generated at the output connector. Note that by selecting an active sequence, you do not automatically change the output to sequenced mode.
seq:def 1,1,10,0,1	This command defines a link. The parameters are (from left to right) link number, segment number, loop counter, advance flag, and sync flag. These parameters are explained in the Generating Sequenced Waveforms section.

Using the Sequence Define command repetitively, you may program a complete definition of your sequence. When entering a large number of links, efficiency can be improved by using an alternate syntax which allows a table of sequence definitions to be downloaded directly.

Use the following command to program a complete table from an array:

seq:data#<array>	This will program the entire sequence table without programming individual links.

The sequence generator steps through the link list in descending order. In Continuous Run mode, the sequence repeats automatically after the last step has been completed. When the generator is in Triggered mode, the output stops at the last point of the last waveform in the sequence. In Gated mode, the sequence always completes after the gate stop signal.

To remove a link from the sequence table, use the following command:

seq:del <n>	This deletes a link from a sequence table, where <n> is the step number to be removed.

To delete the entire sequence table, use the following command:

seq:del:all	This deletes the entire sequence table.

CAUTION

The *seq:del:all* command erases the entire sequence table. There is no undo operation available for this command.

Controlling the Sequence Advance Modes

Use the following commands to control how the sequence advances through the sequence links:

seq:adv auto This specifies continuous advance, where the generator steps continuously to the end of the sequence table and then repeats the sequence from the beginning. For example, if a sequence is made of three segments, 1, 2, and 3, and AUTO mode is used, the sequence will proceed: 1, 2, 3, 1, 2, 3, 1, 2, 3…, with the duration of the loop depending upon the loop counter specified in the sequence table.

seq:adv trig This specifies that the 3172-W6 idles between links until it senses a valid trigger event. This mode is available only when the 3172-W6 is in Triggered Run mode. An attempt to select this mode when the 3172-W6 is in Continuous Run mode will generate a settings conflict error. After a trigger, the generator output resumes until it is once again between links. Then, the output level idles at a DC level equal to the last point of the last generated waveform. If loops (repeats) were programmed, the segment is repeated n times automatically before it begins idling. After execution of all of the programmed loops, the sequencer steps to the next segment in the sequence upon receipt of the next valid trigger event.

seq:adv step This Stepped Advance mode specifies that the sequence advances to the next link only when a valid trigger event has been received. In this mode, the 3172-W6 generates the first segment continuously until a trigger signal advances the sequence to the next segment. If repeats are specified in the sequence table, they are ignored in Stepped Advance mode. Note that this mode requires that the run mode be set to Continuous.

seq:adv mix In this mode, advancing to the next link is controlled by the Advance bit in the link definition. “0” will cause the link to advance automatically to the next link. “1” will cause the link to repeat itself continuously until a valid trigger event has been received, and then the generator will begin executing the next link. Note that this mode requires that the run mode be set to Continuous.

Generating Modulated Waveforms

The modulation generator is a separate instrument within the 3172-W6. Based on DDS technology, it has a wide dynamic range and high linearity throughout the modulation range.

The 3172-W6 can modulate in the frequency, amplitude, and phase domains. When the modulation output is selected but modulation is turned off, the instrument generates a continuous wave (CW) signal, or steady-state sine wave. The following commands control the modulation of the carrier wave:

mod:type off This disables modulation so that the output generates a CW signal. CW is the sine waveform that is being modulated. When placed in Modulation Off, the sine waveform is continuously generated from the main output. In this mode, sine waveforms can be generated from 100 μHz to 30 MHz. Modulation off operates in Continuous Mode only. The CW settings do not automatically change when you switch from one modulation function to another.

mod:type am This selects amplitude modulation (AM). The modulating signal is internal, and the following parameters control the AM scheme: modulation shape, modulation frequency, and modulation depth.

mod:type fm This selects frequency modulation (FM). The modulating signal is internal, and the following parameters control the FM scheme: modulation shape, modulation frequency, and marker placement.

mod:type swe This selects sweep modulation. The modulating signal is internal, and the following parameters control the sweep: start and stop frequency, sweep time and direction, sweep spacing, and marker placement.

mod:type fsk This selects frequency shift keying (FSK). The shift sequence is created in a data table that can hold up to 4,000 frequency shift steps. The following parameters control FSK modulation: shifted frequency, baud, shift data array, and marker placement.

mod:type ask This selects amplitude shift keying (ASK) modulation. The shift sequence is created in a data table that can hold up to 1,000 amplitude-shift steps. The following parameters control ASK modulation: shifted amplitude, baud, shift data array, and marker placement.

mod:type psk This selects the phase shift keying (PSK). The shift sequence is created in a data table that can hold up to 4,000 shift steps. The following parameters control PSK modulation: shifted phase, baud, shift data array, and marker placement.

mod:type fhop This selects the frequency hop modulation. The frequency hop sequence is created in a data table that can hold up to 5,000 frequency hops. The following parameters control frequency hop modulation: dwell mode, dwell time, frequency data list, and marker placement.

mod:type ahop This selects the amplitude hop modulation. The amplitude hop sequence is created in a data table that can hold up to 5,000 amplitude hops. The following parameters control amplitude hop modulation: dwell mode, dwell time, amplitude data list, and marker placement.

mod:type 3d This selects 3D modulation. This is a special mode that modulates frequency, amplitude, and phase simultaneously. You may set the modulation profile externally through applications such as ArbConnection.

Modulation Parameters

The previous section details the modulation schemes and lists the parameters that control the modulating signals. A complete listing of the modulation control parameters is given in the Programming Reference in this manual.

To program the sweep parameters, use the following commands:

swe:star <value> Set the starting frequency for the sweep.

swe:stop <value> Set the ending frequency for the sweep.

swe:time <value> Set the amount of time that will elapse from the start to the end of the sweep.

swe:dir up Set the sweep direction to “up” (from the start frequency to the stop frequency).

swe:dir down Set the sweep direction to “down” (from the stop frequency to the start frequency).

swe:spac lin Select linear sweep steps, where the generator steps the frequency through the sweep range in linear increments.

swe:spac log Select logarithmic sweep steps, where the generator steps the frequency through the sweep range in logarithmic increments.

swe:mark <value> Define marker position. The marker will generate a pulse at the SYNC output when the marker frequency setting is crossed.

Controlling the Carrier Frequency

In general, when you select a modulation scheme, the waveform being modulated (the carrier) is always a sine wave. When you select the modulation function but set the modulation type to “Off”, the output generates an un-modulated, continuous waveform (CW) signal. The frequency setting of the carrier in modulation mode is not the same as for standard waveform mode and must be programmed separately. Use the following command to program the carrier frequency:

mod:carr <value> Set the CW frequency in units of Hz. The same value will be used for all modulation functions.

Controlling the Carrier Base Line

As explained above, the Advanced Trigger mode allows the 3172-W6 output to “idle” when it has finished a waveform segment and is waiting for the next trigger event. The output signal during this time is called the *baseline*.

The 3172-W6 offers two options for the base line:

1) Carrier (un-modulated, CW carrier)
2) DC Level

Use the following command to control the carrier base line:

mod:carr:bas carr Selects continuous wave (CW) when the modulated function idles between trigger events.

mod:carr:bas dc Selects continuous DC level when the modulated function idles between trigger events.

Generating Half Cycle Waveforms

The Half-Cycle function is a special case of standard waveforms, except that the waveforms are generated a half cycle at a time and displaced by a programmable delay time. In continuous mode, the half cycles are generated continuously. In triggered mode, one half at a time is generated only after a valid trigger event is received. There are three half-cycle waveforms that can be generated: Sine, Triangle, and Square. Use the commands below to select the half-cycle function and program the parameters.

func:mod half Selects the half-cycle function. If you have not changed parameters, then the output will generate half-cycle sine waveforms where the halves are separated by 1 μs delay intervals.

From this point you can change one or more of the half cycle parameters just as they would be programmed for the standard waveform generator. Use the following commands to select one of the half cycle waveforms:

half:shap sin	Selects the sine waveform to be generated using the half cycle function.
half:shap tri	Selects the triangular waveform to be generated using the half cycle function.
half:shap squ	Selects the square waveform to be generated using the half cycle function.

After you select the function and waveform, you may program other parameters to adjust the waveform specifically for your application. You may adjust the start phase for the sine and triangular waveforms, or the duty cycle for the square waveform, and you may program the amount of delay between the half cycles.

Chapter 5 contains programming references that will allow you to program all of the half cycle parameters.

Using the Counter/Timer

You may use the 3172-W6 as a counter/timer instrument. When using this function, you may select the measurement function, gate time, and trigger level, and then hold the measurement until you require a reading. The reading is then taken and passed to the host computer for processing.

The 3172-W6 cannot perform as a counter/timer and generate waveforms at the same time. When placed in counter/timer mode, all waveform patterns are purged from the waveform memory, and the 3172-W6 can be used only for measurements.

The counter/timer function provides a means of measuring frequency and timing characteristics of external signals. Use the commands given below to select the counter/timer mode and set up a measurement function.

func:mod coun	Selects the Counter/Timer allowing frequency and time measurements on external signals.

Once you have selected the Counter/Timer mode, you may select the specific measurement function. Available functions include:

- Frequency
- Period
- Period averaged
- Pulse Width

- Totalize (counts the number of trigger events)

Use one of the following commands to select the measurement function:

coun:func freq	Selects the frequency measurement function. The 3172-W6 takes readings continuously and places them in the output queue, waiting for a read operation to clear the queue for the next reading.
coun:func per	Selects the period measurement function.
coun:func aper	Selects the averaged period measurement function.
coun:func puls	Selects the pulse width measurement function.
coun:func tot	Selects the totalize function. The counter will detect and count all trigger events from the trigger input.

You may adjust the gate time and display mode. If you want to take continuous counter readings, use the default display mode as follows:

coun:disp:mod norm	Enables continuous measurements and read cycles.
coun:disp:mod hold	Stops the measurement cycle and performs a single measurement when triggered by a read operation. The next measurement cycle may be performed after you clear the counter buffer using the following command:
coun:res	Resets the counter, clears the output queue, and arms the counter for its next measurement event.

Chapter 5 contains programming references that will allow you to program all of the counter/timer parameters.

Counter/Timer Limitations

A summary of counter/timer limitations is given below.

1. *Measurement speed*

The rate at which the counter performs its measurements depends upon the display mode setting. The Normal setting simulates the display of a bench-top instrument, where the user sees the result of each measurement as it completes. The display time is roughly 300 ms, allowing enough time to check the result after each gate time cycle. The maximum rate is three measurements per second when using low-period gate times. The Hold display mode allows one reading at a time. The reading starts when the input senses a valid trigger signal, and ends after the gate has closed. Processing time for the reading and the display is roughly 100 ms. In this mode, the counter can take a maximum of ten readings per second.

2. *Gate time period must be higher than the signal period*

The gate must open for an interval that allows enough transitions to pass through the counter gate. If the gate time is too short to measure a signal, the gate will open, but no results can be obtained.

3. *Auxiliary functions disable waveform generation*

When the Counter/Timer mode is selected, all operations of the waveform generator are stopped, and the waveform memory is purged.

Synchronizing through the Local Bus

Use the following commands to set up and control synchronization (phase locking) of two modules through the Local Bus:

inst:coup:path lbus This sets the coupling (synchronization) path to the Local Bus (LBUS). This command must be sent to the master module and the slave module(s).

inst:coup:mode mast Send this command to the 3172 that will be the master module to which other modules will be phase-locked.

inst:coup:mode slav Send this command to the slave module(s) that will be phase locked to the master module.

inst:coup:stat on Send this command to the master module, and then to the slave module. It enables coupling (phase locking).

Example:

The following procedure sets up a 3172 module as the master, sets up another 3172 module as the slave, and monitors the outputs of both modules to observe that they are phase locked.

1. Install Local Bus jumpers on both 3172 modules (refer to Chapter 2, in the Local Bus Configuration section).
2. Install the two modules into adjacent slots in a VXI chassis. The 3172 on the left will be the master, and the 3172 on the right will be the slave.
3. Turn on power to the VXI chassis.
4. Connect the output connector of the left module (master) to channel 1 of the oscilloscope.
5. Connect the output connector of the right module (slave) to channel 2 of the oscilloscope.
6. Send the following commands to the left module (the master):

   inst:coup:mode mast
   inst:coup:path lbus
   inst:coup:stat on
   outp on

This configures the module as a master, selects the Local Bus (LBUS) as the coupling path, turns coupling on, and then turns on the output. At this point, the oscilloscope (channel 1) will show that the master module is generating a waveform. Since the frequency and amplitude were not commanded, the waveform will have the default amplitude (5VP-P) and the default frequency (1MHz). The master module is also sending the synchronization signals onto the LBUS.

7. Send the following command sequence to the right module (slave):

   inst:coup:path lbus
   inst:coup:mode slav

inst:coup:stat on
outp on

This selects LBUS as the coupling path for phase locking, sets up the module as a slave to be phase-locked to the master, turns coupling on (enables phase locking), and then turns on the output.

After the above steps have been performed, channel 2 on the oscilloscope will show that the slave module is generating the default waveform: a 5Vp-p sine wave. By comparing the displays of channels 1 and 2 on the oscilloscope, it is verified that the two 3172 units are in-phase.

Controlling P2 Pulse Modes and Parameters

Description from this point pertains to the P2 module only. General operation of the 3172, regardless of whether W6 or P2 modules are installed in it, is given at the beginning of this chapter and separate sections in this chapter describe the operation of either module. Note that the P2 module consists of two independent pulse generators and, therefore, make sure that the proper channel is selected for programming before commands are sent to the module. Channel is selected using the *inst:sel* command. This command is described in detail in the programming section.

Generating Pulse Waveforms

Upon reset, the default waveform type is set to normal pulse. The pulse waveforms and parameters are generated in an analog fashion but are digitally controlled to achieve maximum resolution, accuracy, and stability.

The pulse generator command interface provides a means of specifying pulse parameters in units of time, exactly as would be specified with a bench-top analog pulse generator. Use the instructions below to access and program the pulse parameters.

After a power-on reset, the pulse generator will have the following default settings:

Pulse Mode = Normal

Period = 1 ms

Pulse Width = 100 ns

Amplitude = 10 Vp-p

Offset = 0 V

Polarity = Normal

Transitions = Fast

The pulse generator commands provide access to all pulse parameters just as they would be programmed on an analog pulse generator. Adjusting the pulse shape to the required characteristics requires adjustment of one or more time and amplitude parameters. The 3172-P2 command interface provides all the necessary controls for making the adjustments. Some setting conflicts can occur in situations where parameter ranges are dependent on the settings of other parameters. These setting conflicts will be discussed later in this chapter. Below is a list of all pulse parameters that are accessible using the pulse command interface.

Note

Please be mindful of the possibility of settings conflicts which can affect your ability to get the desired output. For example, the specification of a pulse width that is longer than the pulse period will cause a settings conflict. Therefore, always program the period first and enter other pulse parameters in a descending order. A list of setting conflicts is given later in this chapter.

Selecting a Pulse Mode

The 3172-P2 has five basic pulse modes: Normal, Delayed, Double, Hold Duty Cycle and External Width Control. Use the following commands to select the pulse modes:

puls:mod norm — The 3172-P2 outputs the normal pulse waveform. The parameters that control the pulse shape are period, width, polarity and transitions.

puls:mod del — The 3172-P2 outputs the normal pulse but delayed from the sync output by a pre-determined delay time. Use the *puls:del* command to program the delay value.

puls:mod doub — The 3172-P2 outputs double pulses. The second pulse is delayed from the first pulse by a pre-programmed delay time. Use the *puls:doub:del* command to program the delay value.

puls:mod hold — The 3172-P2 generates a normal pulse waveform with a fixed ratio of pulse width to period, regardless of the period setting. Use the *puls:dcyc* command to program the duty cycle value.

puls:mod ewid — The width control of the 3172-P2 output pulse is referenced to an external input where signals crossing a certain threshold level determine the width of the pulse. This function can also be used for amplitude to pulse width conversion.

Single Pulse Mode

The basic pulse mode is the Single Pulse. Single pulse defines the shape of a single pulse only. In continuous operating mode it appears as a string of pulses with constant period, width and amplitude. In triggered single pulse mode, one pulse is initiated per trigger.

The parameters associated with the basic configuration of single pulse mode are: Period, Width, High and low Levels. These are discussed below. With more complex settings, you can modify pulse polarity, select linear transitions, and define fixed duty cycles.

Figure 3-3 shows a typical real-life single pulse shape and highlights all of its relevant parameters. While most of the parameters shown in Figure 3-3 can be programmed and adjusted for a specific application, some characteristics of the pulse are derived from the quality of the generator and its output stage. These are discussed in the specifications which can be found in Appendix A.

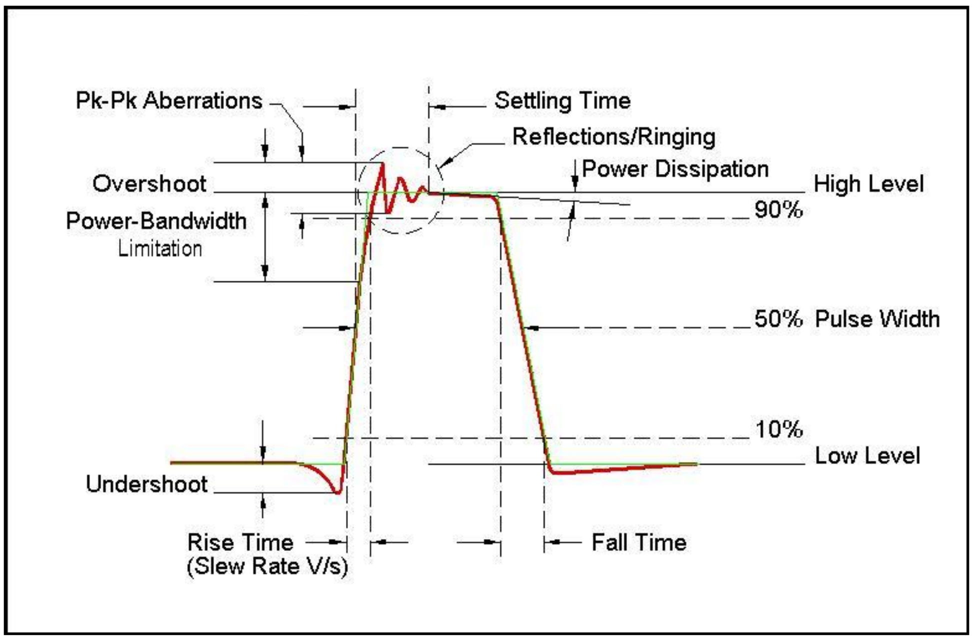

Figure 3-3, Single Pulse Parameters Summary

Period

The period parameter specifies the repetition rate of the pulse in continuous run mode. The period parameter has no meaning if the 3172-P2 is set up to operate in triggered or counted burst run modes. Use the following command to program the period:

puls:per <value>

The period is programmable from 20 ns to 10 seconds. The default value is 1 ms.

Width

The Width parameter specifies the width of the pulse at the 50% point between its high and low level settings. The pulse width interval is not affected by the setting of other parameters such as rise and fall time.

Use the following command to program the pulse width:

puls:wid <value>

The pulse width is programmable from 7 ns to 10 seconds. The default value is 100 ns.

High Level

The high level parameter defines the top amplitude level of the pulse. Any value is acceptable as long as it is larger than the low level setting, does not exceed +20 V and does not fall short of the 5 mV minimum high to low level setting. Use the following command to program the pulse high level:

Volt:hil <value>

The high level is range dependable; it can be programmable from -19.995 V to 20 V. The default value is 2.5 V.

Low Level

The low level parameter defines the lowest amplitude level of the pulse. Any value is acceptable as long as it is smaller than the high level setting, does not exceed -16 V, and does meets the 5 mV minimum high to low level setting requirement. Use the following command to program the pulse high level:

Volt:lol <value>

The high level is range dependable; it can be programmable from -16 V to 19.995 V. The default value is -2.5 V.

Delayed Pulse Mode

Delayed Pulse mode is a special mode that delays the pulse output after a trigger is issued. To select delayed pulse mode, refer to the Selecting a Pulse Mode section above.

The parameters associated with the basic delayed pulse mode are: Period, Width, High and Low Levels, and Pulse Delay. The pulse delay is measured from the trigger edge to the 50% amplitude point of the pulse leading edge, as shown in Figure 3-4.

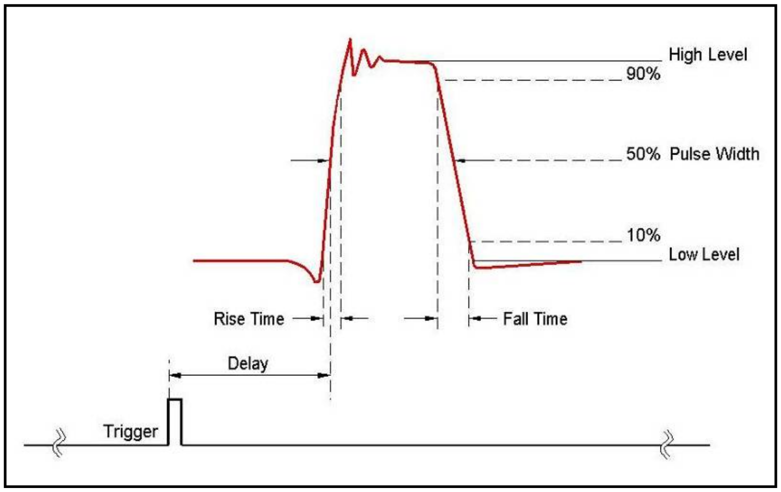

Figure 3-4, Delayed Pulse Mode

Period
The period defines the repetition rate of the pulse in continuous run mode. The period parameter has no meaning if the 3172-P2 is in triggered or counted burst run mode.

Use the following command to program the period:

puls:per <value>

The period is programmable from 20 ns to 5 seconds. The default value is 1 ms.

Width
The Width parameter defines the width of the pulse at the 50% point between its high and low level setting. The pulse width interval is not affected by settings of other parameters such as rise and fall times.

Use the following command to program the pulse width:

puls:wid <value>

Pulse width is programmable from 7 ns to 5 seconds. The default value is 100 ns.

High Level
The high level parameter defines the top amplitude level of the pulse. Any value is acceptable as long as it is larger than the low level

setting, does not exceed +20 V, and exceeds the 5 mV minimum high to low level setting.

Use the following command to program the pulse high level:

volt:hil <value>

The high level is range independent; therefore, it can be programmed from -19.995 V to 20 V. The default value is 2.5 V.

Low Level

The low level parameter defines the lowest amplitude level of the pulse. Any value is acceptable as long as it is smaller than the high level setting, does not exceed -16 V, and exceeds the 5 mV minimum high to low level setting.

Use the following command to program the pulse high level:

volt:lol <value>

The high level is range independent; it can be programmable from -16 V to 19.995 V. The default value is -2.5 V.

Delay

The delay parameter specifies the time between the SYNC output and the first pulse transition. Any value is acceptable as long as it is smaller than the period setting plus the pulse width setting. For longer delays, use the trigger delay function either alone or combined with the pulse delay function for maximum delay resolution.

Use the following command to program the delay value:

puls:del <value>

Pulse delay is programmable from 0 to 5 s.

Double Pulse Mode

Double Pulse mode is a special mode that outputs a pair of pulses at a time. In continuous run mode, the output appears as a series of pulse pairs separated by a time interval set by the double delay time parameter. If you place the instrument in triggered run mode, a pair of pulses is initiated with every trigger. To modify the pulse mode to double pulse mode, refer to the Selecting a Pulse Mode section above.

The parameters associated with the basic double pulse mode are: Period, Width, High and Low Levels and Double Pulse Delay, which sets the delay between the pairs of pulses. Double pulse delay is measured at the 50% amplitude points of the leading edges, as shown in Figure 3-5.

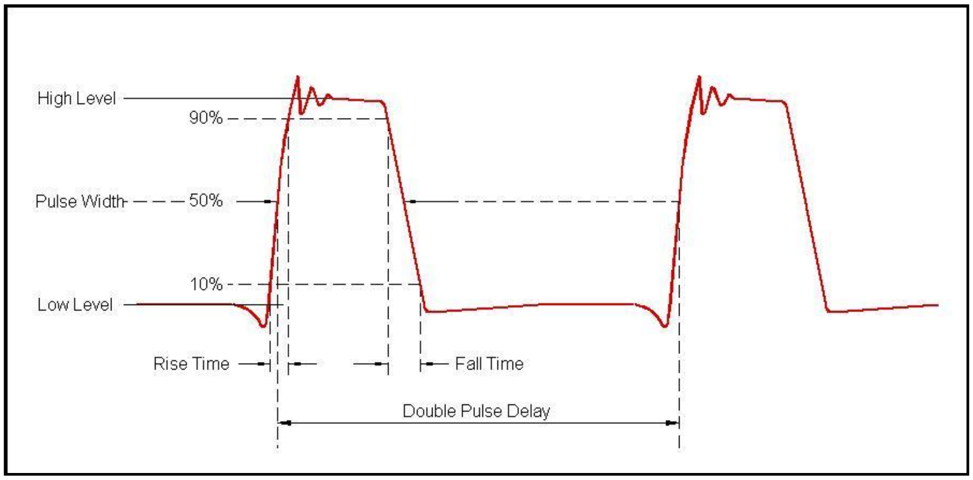

Figure 3-5, Double Pulse Mode

Period

The period parameter specifies the repetition rate of the pulse in continuous run mode. The period parameter has no effect if the 3172-P2 is set up to operate in either the triggered or counted burst run mode.

Use the following command to program the period:

puls:per <value>

The period is programmable from 20 ns to 5 seconds. The default value is 1 ms.

Width

The Width parameter specifies the width of the pulse at the 50% point between its high and low level settings. The pulse width time is not affected by settings of other parameters such as rise and fall times.

Use the following command to program the pulse width:

puls:wid <value>

Pulse width is programmable from 7 ns to 5 seconds. The default value is 100 ns.

High Level

The high level parameter defines the top amplitude level of the pulse. Any value is acceptable as long as it is larger than the low level setting, does not exceed +20 V and does not fall short of the 5 mV minimum high to low level setting. Use the following command to program the pulse high level:

volt:hil <value>

The high level is range independent and can be programmed from -19.995 V to 20 V. The default value is 2.5 V.

Low Level

The low level parameter specifies the lowest amplitude level of the pulse. Any value is acceptable as long as it is smaller than the high level setting, is not below -16 V, and exceeds the 5 mV minimum high to low level setting.

Use the following command to program the pulse high level:

volt:lol <value>

The high level is range independent and can be programmed from -16 V to 19.995 V. The default value is -2.5 V.

Double Pulse Delay

The double pulse delay parameter defines the time that will lapse from the first pulse transition to the paired pulse transition. Any value is acceptable as long as it is smaller than the period plus 2 x pulse width setting. Use the following command to program the double pulse delay value:

puls:doub:del <value>

The delay can be programmable from 0 to 5 s.

Hold DCycle Mode

When single, double, or delayed pulse modes are selected, the programmed pulse width does not normally change when you change the period of the pulse. On the other hand, some applications require that the ratio between the period and the pulse width remain constant regardless of the period setting. In a case like this, use fixed duty cycle mode.

Hold Duty Cycle mode is programmed in units of percent (%) defining the ratio between the pulse width to the period x 100. An example is shown in Figure 3-6. If you program the duty cycle parameter to be 10% and the period to be 50 ms, after you modify the period to 25 ms, the width is adjusted automatically to 2.5 ms so that the duty cycle remains 10%.

To modify the pulse mode to hold duty cycle mode, refer to the Selecting a Pulse Mode section above.

Use the following command to program the duty cycle value:

puls:dcyc <value>

The duty cycle is programmable from 1% to 95% with a resolution of 0.001%. The default value is 50%.

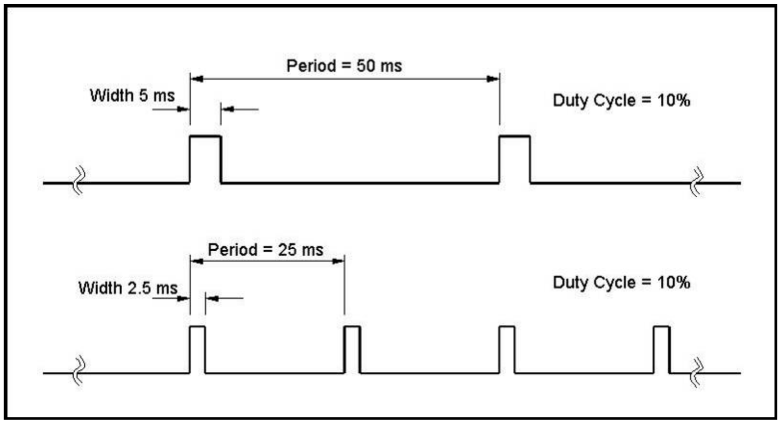

Figure 3-6, Hold Duty Cycle Mode Example

External Pulse Width Mode

External pulse width mode shapes the trigger input and uses it to define the width of the pulses at the pulse output connector. As shown in Figure 3-7, as long as the signal remains below the trigger threshold, the pulse idles at its low level setting. When the signal crosses the trigger level threshold, the output generates the high level with the width of the pulse determined by the time that the signal remains above the trigger level threshold.

To select external pulse width mode, refer to the Selecting a Pulse Mode section above. Note that both the period and width settings are ignored in this mode because both are being set by the external signal. Period and width can be impacted by the trigger threshold and trigger slope settings, as shown in Figures 3-7 and 3-8.

The trigger slope determines if the external pulse width signal's level is above or below the threshold to generate the pulse width. When the trigger slope is set to positive (default), the output will behave as shown in Figure 3-7. Using the same input signal but a negative trigger slope results in an inverted pulse sequence as shown in Figure 3-8.

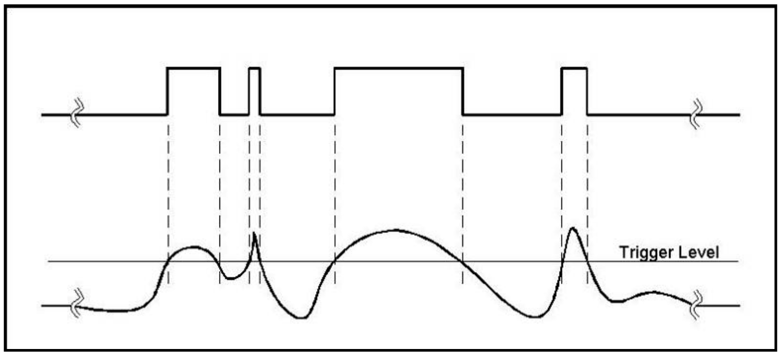

Figure 3-7, External Pulse Width Mode, Positive Slope Example

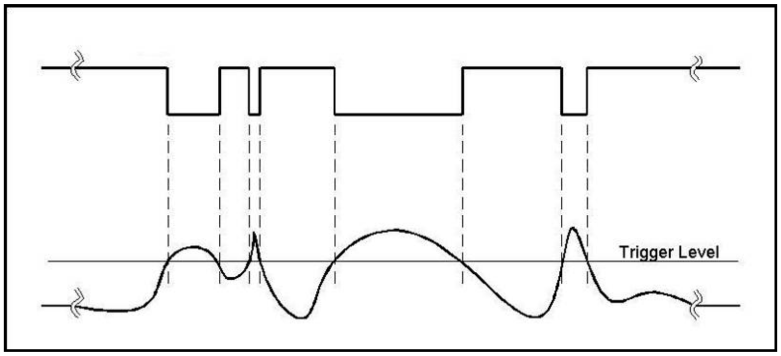

Figure 3-8, External Pulse Width Mode, Negative Slope Example

TIPS

1. **While using external pulse width mode, bear in mind that the period and the width are controlled by a signal that is applied to the trigger input. The trigger input is limited to a width of 10 ns and a frequency of 5 MHz.**
2. **The external pulse width mode does not impose any restrictions on linear transitions, the high and low level settings, or the polarity of the output.**

The external pulse width parameters affect the output only after you select the external pulse width mode.

Use the following command to program the trigger slope:

trig:slop <positive/negative>

The default option is positive.

Programming Pulse Polarity

The pulse polarity parameter determines if the pulse is generated in the Normal, Complemented, or Inverted shape. Pulse polarity can be selected in conjunction with any of the pulse modes except with external pulse width mode. The three polarity modes are available for the symmetrical amplitude range only; for positive and negative amplitude ranges only Normal and Complemented polarity options are available. The various polarity options are shown in Figure 3-9.

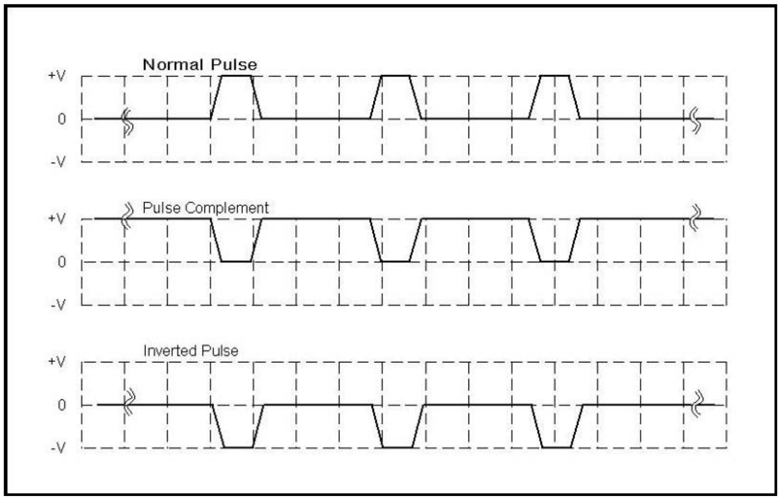

Figure 3-9, Pulse Polarity Options

As shown in this figure, the complemented shape is mirrored around the horizontal axis in a way that the high level becomes low and likewise, the low level becomes high. In complemented mode, the inversion process is symmetrical about the 50% value of the pulse amplitude.

In Inverted mode, the normal pulse is mirrored about the 0 V horizontal axis, positive values are converted to negative and negative values are converted to positive. This mode is not available for negative and positive amplitude ranges because inverting the pulse shape would mean crossing over to other ranges, an operation that will generate setting conflict errors.

The normal pulse and the pulse complement are specifically valuable for applications requiring differential signals and hence a dual channel module like the P2 that has this feature can easily be programmed to generate differential signals. Simply couple both channels and set one channel to Normal and the second channel to Complemented and both signals will be generated differentially, perfectly synchronized, and without phase offset or jitter.

Note that the pulse polarity can be programmed separately for each channel and, therefore, before you modify this parameter, make sure that you program the correct channel.

Use the following command to program the pulse polarity:

puls:pol <normal/complemented/inverted>

The default option is normal.

Applying Linear Transitions

Most of the applications that use pulse generation require that the transitions from low to high and from high to low be done at the fastest possible speed. Such transitions are normally created as a by-product of the output amplifier. General purpose amplifiers that can drive 50 Ω loads with high amplitudes are rare and the products that you can usually buy off the shelf either have poor drive capabilities or uncontrolled aberration capability.

For a pulse generator that generates pulses with fast transitions only, the design problem is simpler because the designer can use a switching amplifier at the output amplifier stage. It takes a very different approach to design an output stage that slows the transitions of the leading and trailing edges. To this extent, the P2 module has a unique output amplifier stage that allows full control over pulse transitions over a very high dynamic range of amplitudes and offsets without degradation of the signal. A comparison between pulses with fast and linear transitions is shown in Figure 3-10. As you can see, the top train has fast transitions; these are normally in the range of <8 ns and are expected to be very fast and without aberrations. The bottom pulse has linear transitions that are expected to have good linearity and slew-rate accuracy. Observe that the pulse width on the pulses that have linear transitions is measured at the 50% amplitude level. Also note that the leading and the trailing edges can be programmed to have different transition times.

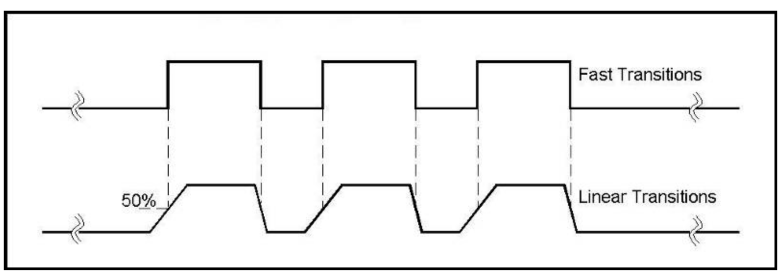

Figure 3-10, Fast and Linear Transitions, Compared

Linear transitions can be used in conjunction with each of the pulse modes, including external pulse width and pulse modulation. Use the following commands to select one of the transition modes:

puls:tran norm The P2 outputs pulses with the fastest transition times. Normal transitions are specified in Appendix A of this manual and are normally measured at less than 8 ns. Normal is the default option for this switch.

puls:tran lin — The P2 outputs pulses with linear transition times. Linear transitions can be programmed separately for the leading and trailing edges within the range of 5 ns to 5 ms. The default value is 10 µs.

puls:tran symm — The P2 outputs pulses with the linear transition times. Linear transitions are programmed simultaneously for the leading and trailing edges within the range of 5 ns to 5 ms. The default value is 10 µs

After you select the linear transition option, you'll probably want to program the transition times for the leading and trailing edges. There are some considerations to observe before you program the transitions. First, note the limits, as specified in Appendix A. These tell you that you can set transition times within the range of 5 ns to 5 ms. Then you have to make sure that the transition settings do not conflict with the pulse width settings. For example, if you set the pulse width to 100 ns and the leading edge transition time to 120 ns, the instrument will not allow you to do the change. A list of settings conflicts is given later in this chapter.

Finally, bear in mind that the transition times are programmed in six ranges and, further, both the leading and the trailing edges must be programmed within the same range.

TIP

Setting conflict may occur if the leading and trailing edges are not programmed within the same range. To avoid such an error, the leading edge value ALWAYS sets the range for the trailing edge and, therefore, always program the leading edge first and then program the trailing edge.

Use the following command to program the transition time for the leading edge:

puls:tran <value>

and use the following command to program the transition time for the trailing edge:

puls:tran:tra <value>

The available transition time rages are shown in Figure 3-11. Note that there are six overlapping ranges that you may use. In-range ratio between minimum to maximum values is 20:1, except the first range that has a 10:1 ratio only. Both values for the leading and trailing edges must be placed inside one range only. For example, you may program 37 ns for the leading edge and 480 ns for the trailing edge because both are values within range 2 but if you program 37 ns for the leading edge and 501 ns for the trailing edge, the instrument will issue a setting conflict message and will ignore the setting of the trailing edge.

The trailing and leading edges parameters affect the pulse output only after you select the linear transitions mode.

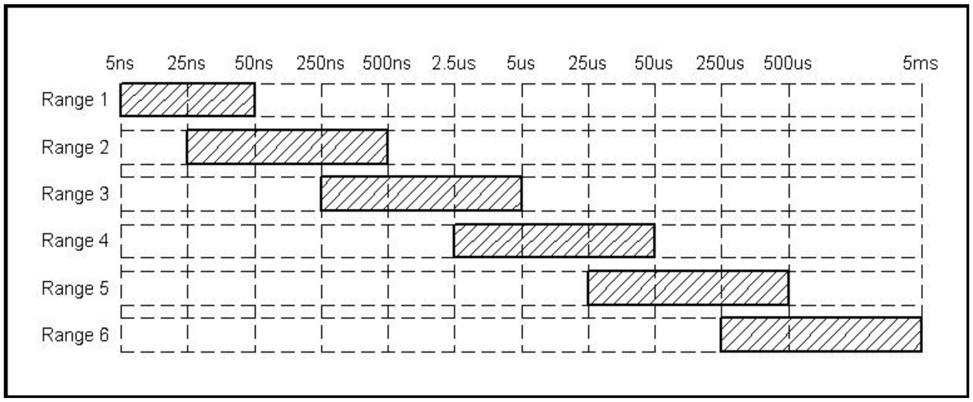

Figure 3-11, Linear Transition Ranges

Pulse Design Limitations

There are a number of limitations that must be observed before you program pulse parameters. These limitations are grouped into two groups: Timing limitations and amplitude limitations. The limitations arise from fact that the instrument has a finite frequency range and a finite amplitude span. These limitations are summarized in Appendix A of this manual. For example, the maximum peak to peak amplitude span for a given pulse design is 22 V (20 V in POS and NEG modes) and a further limitation is that the positive and negative settings cannot exceed the output amplifier rails of +11 V and -11 V (SYMM mode). Another example is the leading and trailing edge transition times. This is limited by design to 5 ns thus faster transitions cannot be programmed.

Besides design limitations, one may enter into a conflicting situation where one parameter exceeds its limits when programmed in conjunction with another parameter. For example, programming a period of 10 ms and pulse width of 100 ms is not possible because, by definition, the pulse width must be smaller than the period. The paragraphs below summarize possible settings conflicts and suggests options to resolve the settings conflict. If you try to program a parameter that will cause a setting conflict, the instrument will automatically detect the problem and issue an error message. In this case, the output may appear distorted and generate uncontrolled signals.

Pulse Setting Errors

Pulse settings errors may involve one or more parameters and there is also a chance that more than one error is embedded in the settings. For simplicity, the P2 generates an error code but its output shape remains unchanged. Each error is indicated by a number and short description that provide clues which of the parameters conflict. Detailed description for each of the conflicting settings is given below.

Settings conflict errors may occur when you program parameters that collide with each other or, when programming a certain parameter, throws the entire pulse shape out of its legal and specified boundaries. The error codes are returned individually when you use the *syst:err?* query. The error codes returned have three digits and these are described below including the explanation of how to remove the error(s) by correcting conflicting or out of range parameters.

Note

The following abbreviations were used throughout the following settings conflict descriptions:

PER – Period setting

HIL – High level setting

LOL – Low level setting

WID – Pulse width setting

DEL – Single pulse delay setting

DDEL – Double pulse delay setting

LEE – Leading edge setting

TRE – Trailing edge setting

BUR – Burst count setting

DUTY – Duty cycle setting

-222,"Data out of range"

Error -222,"Data out of range" occurs when attempting to program an amplitude which is smaller or larger than the specified limit. This error will occur in all run mode options and under the following conditions:

HIL – LOL ≤ 2.5 mV; HIL – LOL ≥ 22 V (20 V for POS/NEG ranges)

This error will be detected on all pulse modes and options. The minimum level is an absolute value that the P2 can accept. The same will occur if you reverse the high and low levels because the instrument will sense it as a negative voltage which is less than the minimum 2.5 mV limit.

The low and high limit errors are detected by a mechanism that is common to all of the operating modes of the instrument and is not just unique to the operation of the pulse generator.

Corrective Actions

1. Modify the high level value
2. Modify the low level value

-500,"Transitions Exceed Width"

Error -500,"Transitions Larger than Width" occurs when attempting to program leading or trailing edge values that are larger than the programmed pulse width. This error may occur in all pulse and run modes except in external pulse width pulse mode, under the following conditions:

0.625(LEE + TRE) ≥ WID

When such an error occurs, the resultant output would have looked as illustrated in Figure 3-12 (red line). To correct the problem and to restore the pulse generator to normal operation, use one or more of the corrective action options listed.

Corrective Actions

1. Increase the pulse width value
2. Decrease the leading edge value
3. Decrease the trailing edge value

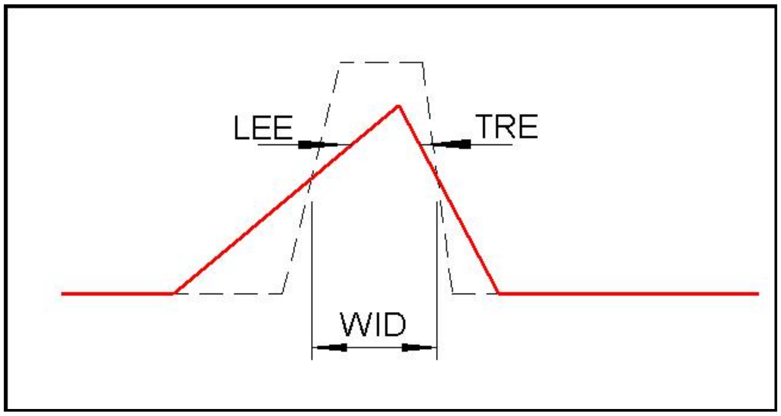

Figure 3-12, Output Waveform with Error -500 Example

-501,"PW Plus Transitions Exceed Width"

Error -500,"PW plus Transitions Exceed Width" occurs when attempting to program a pulse width value that is larger than the programmed pulse period. This error may occur in single pulse mode only and in continuous run mode and under the following conditions:

WID + 0.625(LEE + TRE) ≥ PER

When such an error occurs, the resultant output would have looked as shown in Figure 3-12 (red line). To correct the problem and to restore the pulse generator to normal operation, use one or more of the corrective action options listed.

Corrective Actions

1. Decrease the pulse width value
2. Increase the pulse period value, or
3. Decrease the leading edge value
4. Decrease the trailing edge value

-502,"Transitions Outside of Range"

Error -502,"Transitions Outside of Range" occurs when attempting to program the trailing edge value outside of the leading edge range. This error may occur in all pulse modes except external pulse width and in all run modes under the following conditions:

LEE & TRE ≠ In Range

To correct the problem and to restore the pulse generator to normal operation, use one or more of the corrective action options listed.

Corrective Actions

1. Re-program the leading and trailing edge values to be within the same range as shown in Figure 3-11.

-503,"Double Delay Conflict"

Error -503,"Double Delay Conflict" occurs when attempting to program leading or trailing edge values that are larger than the programmed double pulse delay. This error may occur in double pulse mode only and in conjunction of all run modes under the following conditions:

WID + 0.625(LEE + TRE) ≥ DDEL

When such an error occurs, the resultant output would have looked as illustrated in Figure 3-13 (red line). To correct the problem and to restore the pulse generator to normal operation, use one or more of the corrective action options listed.

Corrective Actions

1. Decrease the pulse width value
2. Increase the double delay value
3. Decrease the leading edge value
4. Decrease the trailing edge value

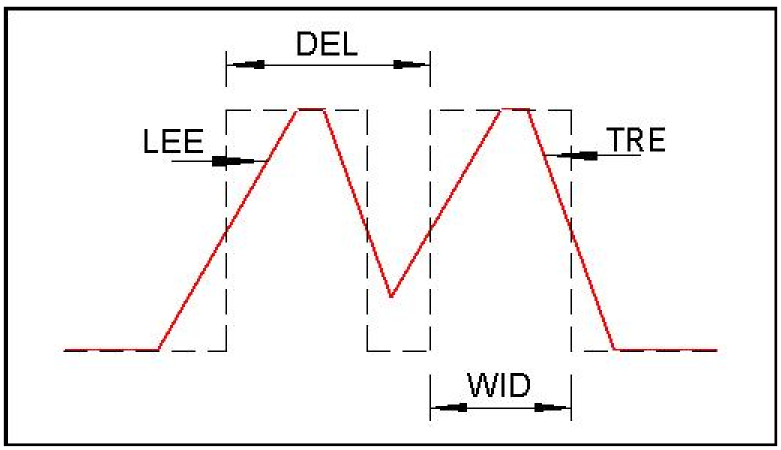

Figure 3-13, Output Waveform with Error -503 Example

-504,"Double Delay and Width Conflict"

Error -504,"Double Delay and Width Conflict" occurs when attempting to program a double delay to a pulse width value that is larger than the programmed pulse period. This error may occur in double pulse mode and in continuous run mode under the following conditions::

DDEL + WID + 0.625(LEE + TRE) + 4 ns ≥ PER

When such an error occurs, the output would have looked as illustrated in Figure 3-13 (red line). To correct the problem and to restore the pulse generator to normal operation, use one or more of the corrective action options listed.

Corrective Actions

1. Increase the double delay value
2. Decrease the pulse width value

3. Increase the pulse period value, or
4. Decrease the leading edge value
5. Decrease the trailing edge value

-505,"Delay and Width Conflict"

Error -505,"Delay and Width Conflict" occurs when attempting to program a delay to the pulse width value that is larger than the programmed pulse period. This error may occur in delayed pulse mode and in continuous run mode under the following conditions:

DEL + WID + 0.625(LEE + TRE) + 4 ns ≥ PER

When such an error occurs, the resultant output would have looked as illustrated in Figure 3-12 (red line). To correct the problem and to restore the pulse generator to normal operation, use one or more of the corrective action options listed.

Corrective Actions

1. Decrease the delay value
2. Decrease the pulse width value
3. Increase the pulse period value, or
4. Decrease the leading edge value
5. Decrease the trailing edge value

-506,"Duty Cycle not in Range"

Error -506,"Duty Cycle not in Range" occurs when attempting to program duty cycle value below the minimum pulse width range. This error may occur in half duty cycle pulse mode only and in conjunction with any of the run modes under the following conditions:

DUTY CYCLE < WID Range

To correct the problem and to restore the pulse generator to normal operation, use one or more of the corrective action options listed.

Corrective Actions

1. Increase the duty cycle value
2. Increase the period value

-507,"Delay Conflict"

Error -507,"Delay Conflict" occurs when attempting to program a leading or trailing edge transition that is larger than the programmed pulse delay. This error may occur in delayed pulse mode only and in conjunction with all run modes under the following conditions:

WID + 0.625(LEE + TRE) ≥ DEL

When such an error occurs, the resultant output could look as illustrated in Figure 3-13 (red line). To correct the problem and to restore the pulse generator to normal operation, use one or more of the corrective action options listed.

Corrective Actions

1. Decrease the pulse width value
2. Decrease the delay value
3. Decrease the leading edge value
4. Decrease the trailing edge value

Controlling A3 Output

The A3 module is a single-channel, high-voltage amplifier. It does not generate waveforms itself, but simply amplifies low-voltage signals from a W6P2 module. To use it with the W6 waveform generator module, use a coaxial cable to connect the W6 output to the A3 input. You can then command the A3 module to either amplify the W6 signal or simply pass it through to its output connector, bypassing the amplifier.

When the A3 module is set to bypass the W6 output signal, the signal still retains its full bandwidth. However, when the output amplifier is engaged, the amplifier limits the bandwidth to 100 kHz.

Upon reset, the A3 defaults to the OFF condition, in which it bypasses the amplifier and routes the W6 signal routed directly to the output connector.

The amplifier is the ON condition using the following command:

outp:ampl 1

This activates the amplifier circuits. However, the toggle switch on the A3 front panel must be set to the ON position to apply the amplified signal to the output connector. For safety purposes, a protective cover prevents inadvertent activation of the toggle switch.

The A3 has two operating modes:

- High: The output swing can reach 122Vp-p
- Low: The output is limited to 61Vp-p.

A built-in mechanism limits the voltage of the power supply rails on the amplifier circuit when low-amplitude signals are generated. This prevents the amplifier from overheating. If your output swings from low to very high voltages, you may allow the A3 to select the rail voltages automatically. However, if you are operating in a specific range, it is recommended that you set up the supply voltages correctly.

Use the following command to generate signal swings up to 122Vp-p:

outp:ampl:pow high

For low output levels use the following command:

outp:ampl:pow low

Finally, to let the A3 automatically select the best power supply *voltages, use the following command:*

outp:ampl:pow auto

Warning

The A3 module is equipped with a hazard-protection toggle switch on the front panel. High voltages are not present at the output connector when the switch is in the OFF position. You may use this switch to disable high-voltage at the output in case of emergency.

Never touch the center pin of the output connector with bare hands or with a metallic object as lethal voltages may exist on this pin.

Always exercise caution when connecting a cable to the A3 output. Make sure the high power switch is in the OFF position while connecting a cable to the A3 output connector.

This page was left intentionally blank.

Chapter 4
ArbConnection

What's in this Chapter?

This chapter explains how to install, invoke, and use the ArbConnection application. It provides instructions for programming instrument controls and parameters, creating waveforms, and downloading waveforms to the 3172.

What is ArbConnection?

ArbConnection (part number 922336-001) is a utility program included with the 3172 CD that aids in controlling the 3172 from a remote computer. It provides three types of functions:

Front panel control. Through a simulated front panel, you can control the 3172 in much the same manner as a bench-top instrument.

Waveform generation and editing. Using drawing tools and equations, you may define and edit arbitrary waveforms.

Downloading. After you define a waveform, you may download it to the waveform memory on the 3172.

System Requirements

To use ArbConnection, you need the following:

1. Computer, Pentium III or better
2. Windows® 2000, Windows XP®, Windows Vista, or Windows 7
3. Screen resolution of at least 1024 x 768 pixels
4. Pointing device, mouse, or trackball
5. National Instruments VISA, version 2.6 or higher, or equivalent

Installing ArbConnection

Before you install ArbConnection, make sure that there is at least 10 megabytes of available space on your hard disk.

Installation of ArbConnection requires the visa32.dll runtime engine. You may download the latest version of this file from National Instruments' web site, **www.ni.com.**

After downloading this file, move it to your Windows system folder. Then run the "setup.exe" file on the ArbConnection installation CD. The installation program installs ArbConnection on a logical drive of your choice (the default is drive C:). It automatically creates a new folder and copies the files that are required to run the program. Then it creates a workgroup and icons to start ArbConnection.

Startup & Communication Options

Invoke ArbConnection by double-clicking the icon on the desktop. If you cannot find the icon on your desktop, click on Start -> Programs -> ArbConnection. The “Startup & Communication Options” dialog box displays as shown in Figure 4-1.

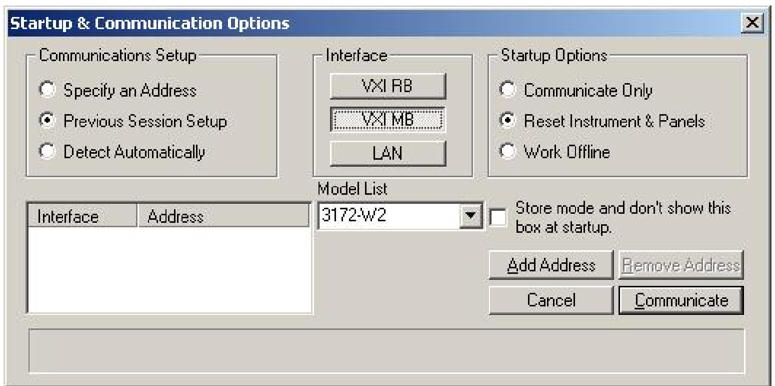

Figure 4-1, Startup & Communication Options Dialog Box

If desired, you may check the “Store mode and don’t show…” box to prevent this dialog box from displaying every time you invoke ArbConnection.

The purpose of this dialog box is to configure the program to communicate properly with the 3172. For example, if you are using a GPIB device that has address 4, you may click “Specify an Address” and then enter the required address. Then, ArbConnection will automatically use the specified address each time it starts up.

If you choose not to have this dialog box displayed automatically at startup, you may still access and change the options from the System command, at the top of the ArbConnection window.

Make your selection and then click “Communicate.” The “Startup & Communication Options” dialog box will close, and the main window displays.

ArbConnection Features

ArbConnection provides complete control over all features of the 3172. Using ArbConnection, you may set up the 3172 to generate waveforms from sources such as the built-in library of standard waveforms, arbitrary waveforms from user-downloaded coordinates, modulated waveforms, and digital patterns. You may also access these features through software utilities such as VXI*Plug&Play* drivers and soft front panels.

Main Window

The main window includes a standard Windows menu bar at the top (Figure 4-2). It provides access to operations such as loading and saving files, setting viewing options, and configuring the 3172.

The Link bar is immediately below the menu bar. The Link bar provides direct access to instruments that are active on the interface bus. ArbConnection can control a number of instruments, such as the Model 3172, simultaneously. If you connect an instrument while ArbConnection is running, ArbConnection automatically detects the instrument and adds its name and associated address to the drop-down list in the Link bar. If you run ArbConnection in offline mode, the Link bar will show “3172, Offline.”

The Panels toolbar is shown in Figure 4-3. By clicking the buttons on the Panels toolbar, you may access the corresponding virtual control panels (detailed later in this chapter). When you launch ArbConnection, the Output panel is initially open.

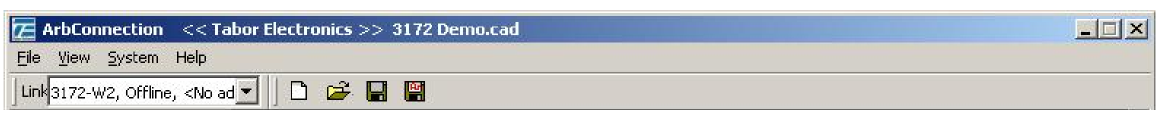

Figure 4-2, ArbConnection Menu and Link Bar

Figure 4-3, Main Panels Toolbar

Control Panels

Each control panel replicates the look and feel of a bench-top instrument’s front panel. Refer to the Output panel in Figure 4-5.

Other panels are similar, so the following description of the Output panel serves as guide for controlling the rest of the panels.

Looking at Figure 4-5, identify the following controls and indicators:

Pushbuttons

LEDs

Radio buttons

Dial

Digital display

The functions of these are as follows:

Pushbuttons – Clicking the mouse on a pushbutton toggles an option on and off. For example, clicking the State button in the Output section turns the 3172 output on. To help indicate this, the button then appears as though pushed in, and a red bar at the center of the button appears to be illuminated. Clicking the Output button a second time turns off the output, and then the button no longer appears pushed in or illuminated.

LEDs – LEDs indicate which of the parameters are displayed on the digital display. A red LED indicates that the parameter name next to this LED is selected. Only one LED can be "on" at a time.

HINT

To turn on an LED, click on the LED or on the text next to it. The selected parameter is then indicated by a darker LED shade.

Radio Buttons – Radio buttons are used for changing operating modes, or selecting between mode options. One of the radio buttons is always "on", with a red dot in its center to indicate its state. These are referred to as "radio buttons" because only one can be on at a time, as with a radio that has preset buttons.

Dial – The dial is a tool for adjusting a number in the display area. To use the dial, point to it with the mouse and then press and hold the left mouse button. While holding the mouse button down, move the mouse in a clockwise circle to increase the displayed number, or counterclockwise to decrease the number. The dial modifies digits at the cursor position, and allows modification within the legal range of the displayed parameter. Once you have reached the end of the range, further dial movement has no effect on the display. You may also change the display reading without the dial by using the [↑], or [↓] keys, or by simply typing the desired number using the computer keyboard.

NOTE

After you change the displayed number, the 3172 will be updated with the new parameter only after you click on the Modify/Execute knob.

Digital Display – The digital display is a tool for displaying various 3172 parameters, just as on a physical control panel.

Note

The normal color of a displayed number is dark blue. If you modify the number, its color changes to a lighter shade of blue, indicating that the 3172 has not been updated yet with the new value. Clicking on the Modify/Execute knob will update the instrument and restore the color of the digital readout to dark blue, indicating that the actual 3172 setting now matches the displayed number.

Also note that the digital readout has an auto-detect mechanism for high and low limits. You cannot exceed the limits when using the dial, but you may if you use the keypad. If you enter a number that exceeds the limits, ArbConnection will not let you update the instrument with the setting until you correct it.

Figure 4-4, Operation Panel Selection

The Operation Panels

The Operation panels provide control over the basic operation of the 3172. From these panels, you may select the output function and run mode, turn the output on and off, and adjust parameters for various functions. There are five panels in this group:

Output

Run Mode

Standard

Arbitrary/Sequence

Half Cycle

The Output panel is always visible because this is the panel that controls operating functions, run modes and sets the outputs on and off. You may hide or show other panels by clicking the appropriate item under “Operation” (Figure 4-4).

The Operation panels are detailed in the next section.

Output

ArbConnection displays the Output panel, shown in Figure 4-5, automatically. The buttons and LEDs are arranged in the following groups:

General Parameters. These controls adjust amplitude and offset.

Wave Mode. This group lets you select the waveform mode.

Run Mode. These controls are for selecting the Continuous mode or one of the interrupted modes (Trigger, Gated, or Burst).

PLL. These controls enable and disable phase locking, and select the source signal.

Sync Output. This group enables and disables the sync signals on the VXIbus backplane and front panel, selects the sync qualifier, and allows you to adjust the sync pulse position and width relative to the waveform.

Output. These controls are for turning the output signal on and off, and for selecting the load impedance.

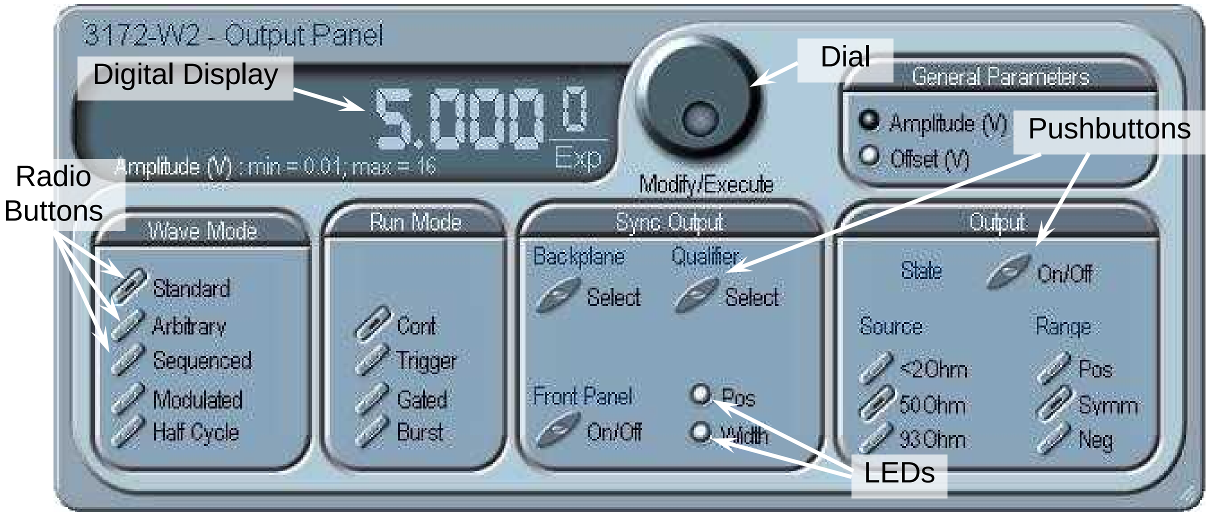

Figure 4-5, Output Panel

When you click on a button, the 3172 responds immediately. When you change a numeric parameter on the display, the 3172 does not respond until you click on the Modify/Execute knob to update the instrument.

Some controls in this panel also appear in other panels. When you change a parameter in this panel, the other panels are updated automatically.

The functional groups listed above are explained in detail below.

General Parameters

The General Parameters group contains two parameters: Amplitude and Offset.

To access a parameter, click on its name. The LED next to the parameter then changes to “on” and the display shows the current value. You may use the dial, keyboard, or [↑] and [↓] keys to adjust the value. After you change the value, click on the Modify/Execute knob to update the 3172.

Wave Mode

The Wave Mode group is used for selecting which of the available waveforms will be generated at the output connector. The 3172 provides five types of waveforms: Standard, Arbitrary, Sequenced, Modulated, and Half Cycle. Click one of these buttons to select the waveform type. The default function type is Standard. If you want to change the Standard waveform parameters, you may select Standard from the Panels bar.

Run Mode

Using the controls in the Run Mode group, you may select Continuous mode or one of the interrupted modes (Triggered, Gated, or Burst). There is no additional panel for Continuous mode, but if you click one of the other run mode options, then you may adjust the trigger parameters from the Trigger panel.

SYNC Output

SYNC Output group has buttons that control the state of the SYNC output and the position and width of the sync pulse relative to the waveform. It also has buttons to control the VXIbus backplane TTLTrg0-7 and ECLTrg1 outputs and the sync validation source. Click on the State buttons to toggle the outputs on and off.

The operation of the SYNC output is explained in Chapter 3. Note that the position parameter affects the output only when placed in BIT, LCOM, or Pulse mode, and the width affects the output only when Pulse mode is selected.

Output

The Output Control group controls the state of the main output only. Click on the State buttons to toggle the outputs on and off.

The load impedance button allows you to calibrate the output amplitude to compensate for the actual load impedance value. The default impedance is 50 Ω, and the output level is adjusted in reference to this value. For lower or higher load impedance setting you may use the <2 Ω or 93 Ω setting. If the correct load impedance is not matched, expect that your amplitude reading might be lower or higher than programmed.

The range button is used for selecting the amplitude range. Symmetrical is the default setting. Using this mode, programmed values will divide the level into two and will generate half of the amplitude setting positive and half of the amplitude setting negative. For example, 5 V amplitude will generate an amplitude span of -2.5 V to +2.5 V. The positive range generates level above 0 V only and the negative range generates negative levels below 0 V only.

Run Mode

To work with the Run Mode Control Panel (Figure 4-6), click on the Run Mode button on the Panels toolbar. Note that when you invoke the Run Mode Control Panel, the trigger mode does not change. To change the run mode, use the Output panel. The trigger parameters and settings in the Run Mode Control Panel take effect only if you have selected an appropriate run mode.

The Run Mode Control Panel has its controls divided into the following groups:

Trigger Modifier
The Trigger Modifier group provides control over the retrigger interval and delay time.

To change the trigger delay or the re-trigger interval, click on one of these parameters. The digital display then shows the current value, which you may adjust using the dial, keyboard, or the [↑] and [↓] keys. After you adjust the value, click on the Modify/Execute knob to update the 3172.

Figure 4-6, Run Mode Control Panel

Trigger Source
The 3172 accepts triggers from a number of sources: Bus, VXI Backplane (TTL Trigger 0 through 7 and ECL Trigger 1), External, and Internal. The VXI backplane trigger lines can synchronize operation with other devices residing in the VXIbus chassis.

The various trigger source options are:

Bus – Disables all trigger inputs and allows software triggers only.

External – Enables the front panel trigger input and disables all other sources.

Internal – Enables an internal, non-synchronized trigger generator, but will not allow triggers from any other sources.

TTLT0-7 – Enables one or more backplane trigger lines. Note that the 3172 can receive triggers from more than one TTLTrg line, but will not accept triggers from bus, external, or internal trigger sources.

ECLT1 – Enables the ECLTrg1 input and will allow trigger events from this backplane line.

Trigger Parameters

Gated Mode: There are two modes that define how the 3172 will gate. The standard mode is Level, in which a trigger signal below the threshold level disables the output, and a trigger signal above the threshold level enables the output. The other mode is Transition, in which each transition toggles the gate on or off. The transition direction is programmable using the Slope options.

Slope: If you click on “Pos”, the instrument triggers on the positive-going (rising) edge of the trigger signal. Similarly, if you click on “Neg”, the instrument will trigger on the negative-going (falling) edge of the trigger signal. Note that this affects only the signals that are accepted from the front panel trigger input.

Burst: Programs the burst counter for burst mode. Once triggered, the 3172 outputs a series of output waveforms that ends when the burst counter reaches the specified count.

Timer: The Timer button lets you set the trigger period of the free-running internal trigger generator. The internal trigger timer is programmed in units of seconds. Note that the internal trigger generator function is available in Pulse mode only. Other output functions use the re-trigger generator, which has a meaning different from that of the internal trigger period. Information on the Re-trigger mode is given in Chapter 3.

Trigger Level: Programs the trigger level parameter. Depending on the slope setting, the 3172 will be triggered to output waveforms when the trigger level threshold has been crossed.

Manual Trigger

The TRIG button operates only in conjunction with the BUS mode. Press the TRIG button to trigger the instrument as if an external trigger has been applied.

Standard

The Standard Waveforms Panel (Figure 4-7), is accessible after you click on the Standard button in the Panels bar. The Standard Waveform Panel groups allow adjustment of channel control, parameters, 10 MHz reference, and waveforms. The functional groups in the Standard Waveforms panel are described below.

General Parameters

The General Parameters group has controls for Amplitude and Offset. The values you set in this panel may be duplicated on other panels, so whenever you change amplitude and offset in the Parameters group, the other panels update automatically.

To access the required parameter, click on the parameter name. The indicator next to the required parameter highlights. The digital display then shows the value associated with the highlighted indicator. Use the dial, keyboard, or the [↑] [↓} keys to adjust the reading to the required setting. After you modify the reading, click on the Modify/Execute knob to update the 3172 with the new reading.

Figure 4-7, Standard Waveforms Panel

Waveforms

The Waveforms group provides access to a library of built-in standard waveforms. The library includes Sine, Triangle, Square, Pulse Ramp, Sinc, Exponential, Gaussian, and DC waveforms. Each waveform has one or more parameters to adjust the required characteristics of the output. For example, phase start can be adjusted for the sine and triangle waveforms, and duty-cycle can be adjusted for the square waveform. For the pulse waveform, you may adjust the rise and fall time, as well as the width and delay. Parameters associated with each waveform are automatically displayed when you select the waveform.

Note that by clicking a button in this group, you immediately update the 3172 output with this waveform shape.

Parameters

The parameters group contains buttons that control the output frequency and the 10 MHz reference source.

The Frequency control lets you set the output frequency of the selected waveform shape. When this control is selected and highlighted, you may modify it using the dial, keyboard, or [↑] [↓} keys to adjust the readout to the required setting. After you modify the reading, click on the Modify/Execute knob to update the 3172 with the new value.

10 MHz Ref – The 10 MHz group contains buttons that control the source of the 10 MHz reference for standard waveforms. The 10 MHz clock is the reference that feeds the sample clock and the DDS clock, and therefore determines accuracy and stability. The internal 10 MHz source has 1 PPM stability over the operating temperature range, and a time stability of 1 PPM per year). The accuracy of the internal source is adjustable, but will shift with time and temperature.

When better accuracy or stability is required, or when clock synchronization to other devices is necessary, you may select another source. The 10 MHz source options are:

Internal: from the built in source

External: applied to the front panel 10 MHz input connector

CLK10: Available on the VXI backplane. The CLK10 source has the least accuracy and stability of the three options, but is useful for synchronization with other VXI modules.

Arbitrary/Sequence

The Arbitrary/Sequence panel (Figure 4-8), is invoked by clicking the Arbitrary/Sequence button on the Panels bar. Note that if you invoke the Arbitrary/Sequence Panel from the Panels menu, the 3172 will not change its output type. On the other hand, if you select the Arbitrary or Sequenced option from the Main Panel, the 3172 will immediately change its output to the selected waveform type. The functional groups in the Arbitrary Waveforms Panel are described below.

General Parameters

The General Parameters group contains two parameters: Amplitude and Offset. The values in this group may be duplicated on other panels. When you change amplitude or offset in the Parameters group, the other panels are updated automatically.

SCLK

The SCLK (Sample Clock) controls let you select the source of the sample clock and set the sample clock frequency. The sample clock setting affects the 3172 in Arbitrary mode only. It is programmed in units of samples per second (S/s), and will affect the instrument only when it is programmed to output arbitrary or sequenced waveforms. The SCLK parameter has no effect on the frequency of standard waveforms.

The three switches in the SCLK group set the sample clock input to Internal, External, or ECLT0. The default is Internal. When you select External, make sure an appropriate signal is connected to the external sample clock connector on the rear panel. The ECLT0 source is a backplane signal that allows ECL level signals to travel to all VXI modules.

Click on the SCLK button to access the SCLK parameter. The value that is associated with the highlighted indicator appears on the digital display. You can use the dial, keyboard, or the [↑] [↓} keys to adjust the SCLK setting. After you modify the setting, click on the Modify/Execute knob to update the 3172 with the new reading.

Parameters

The Parameters group contains three parameters: Amplitude, Offset, and Segment. The amplitude and offset values displayed in this group are the same as in the Main Panel. Whenever you change amplitude or offset in the Parameters group, the Main panel is updated automatically. The Segment parameter provides access to the active segment for each channel.

To access the required parameter, click on the parameter name. The indicator next to the required parameter is then highlighted. The digital display shows the value associated with the highlighted indicator. You may use the dial, keyboard, or the [↑] [↓} keys to adjust the setting. After you modify the setting, click on the Modify/Execute knob to update the 3172 with the new setting.

Figure 4-8, Arbitrary & Sequence Panel

***10 MHz* Ref**

The 10 MHz group contains buttons that select the source of the

10 MHz reference for standard waveform. The 10 MHz clock is the reference that feeds the sample clock and the DDS clock, and therefore determines accuracy and stability. The internal 10 MHz source has 1 PPM stability over the operating temperature range, and time stability of 1 PPM per year. The accuracy of the internal source is adjustable, but will shift with time and ambient temperature.

When better accuracy or stability is required, or when clock synchronization to other devices is necessary, you may select another source. The 10 MHz source options are:

Internal: from the built in source

External: applied to the front panel 10 MHz input connector

CLK10: Available on the VXI backplane. The CLK10 source has the least accuracy and stability of the three options, but is useful for synchronization with other VXI modules.

Sequence

The Sequence Advance Mode group provides control over advanced modes for the sequence generator. Advanced options include Auto, Stepped, Single, and Mixed. Refer to Chapter 3 for details about these advanced modes.

Memory Management

The Memory Management group provides access to the Memory Partition and Waveform Studio Screens. The Waveform Partition button opens a screen as shown in Figure 4-9. The Waveform Studio button opens the screen shown in Figure 4-10. Instructions for using these screens are given in the following paragraphs.

Using the Memory Partition Table

Refer to Chapter 3 for more information about waveform memory and segment control. In general, the 3172 generates arbitrary waveforms, but they must first be downloaded from the host computer to the 3172 waveform memory.

You do not have to use the entire memory when you download a waveform. Model 3172 allows memory segmentation, so that up to 16 k smaller waveforms may be stored in this memory.

There are two ways to divide the waveform memory into segments:

Define a segment and load it with waveform data, define the next segment and load with data, etc.

In ArbConnection, make up one long waveform that contains many smaller segments, download it to the instrument in one operation, and then download a memory partition table that splits the entire waveform memory into the required segment sizes.

To use ArbConnection to download one long waveform and then segment it into smaller sections, follow this procedure:

Click Memory Partition. The dialog box shown in Figure 4-9 appears.

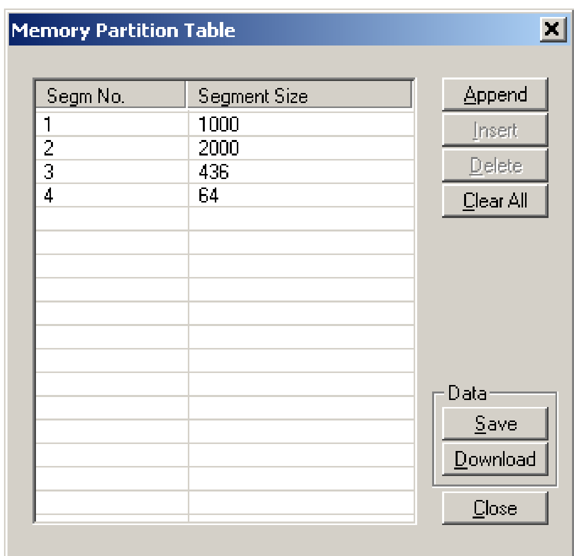

Figure 4-9, Memory Partition Table

The two main fields in the segment table are “Segm No.” (segment number) and “Segment Size”. “Segm No.” is an index field that can have values from 1 to 16 k. The segment size is always associated with the segment number. You may program any segment size from 16 (10 in legacy mode) to the capacity of the memory.

Click on the Append button to add a segment at the end of the segment list. If you click a segment, it will highlight, and the Append button becomes an Insert button. Use the Insert button to insert a segment before the highlighted segment. Use Delete button to delete the highlighted segment.

The Clear All button removes all segments from the table and lets you start a new segment table.

Click on the Close button discard of the contents of the dialog box without saving your last actions, and to remove the Segment Table from the screen.

The Save button saves the current session so that you may continue to configure the Memory Partition table from the same point later on. The Download button updates the 3172 with the present segment table settings.

TIP

The Memory Partition table does not download waveforms. Use the memory partition table only if you have merged multiple waveforms into one. The purpose of the partition table is to divides the memory contents into separate segments, each containing a waveform. If you download waveforms using Waveform Studio, then the memory is already segmented for the waveforms, and there is no need to use the memory partition table.

Using Waveform Studio

Waveform Studio (Figure 4-10), provides access to waveforms that are already stored as files in the host computer. You may download waveforms from such files to various segments in the 3172 waveform memory, and later use them as individual waveforms or combine them into complex sequences.

Waveform Studio has a Segment Table section and a Sequence Table section, as described below.

Segment Table

Using the segment table, you may list and download waveform files that are stored on the host computer. For each waveform, the table shows the segment number, associated file name, length, and download status. You may also download waveforms to memory segments using Wave Composer or individual function calls, but Waveform Studio makes the process easy by combining multiple and complex commands into one simple dialog box.

To access the segment table, click anywhere in the Segment Table area. It will then turn white.

The Segment Table area is divided into three parts: the table area, the waveform shape area, and control buttons. When you click one of the waveforms, the Waveform Shape window displays it.

The Segment Table has four fields:

The “Seg” field contains numbers from 1 through 2,048, designating the programmed memory segment. Note that memory segments are numbered from 1 to 16 k.

The State field shows the current status of the memory segment. It can be *Free*, if no file has yet been assigned to this segment number, or *Mapped*, if file name has been assigned to the segment but the Download button has not been used yet to move the file to the 3172 memory, or *Loaded*, if the process has been completed by pressing either the Download button or the All (download all) button.

The File field is an edit field that lets you browse and select file names to be applied to a specific memory segment. To change or add file name, point and click on the File name field and either type your path or browse to the file location and let Windows find the right path.

The Length field displays the length of the selected memory segment. Memory segments size may be programmed from 16 to the maximum size of your installed memory. Note that the length field is not accessible and shown for reference purpose only.

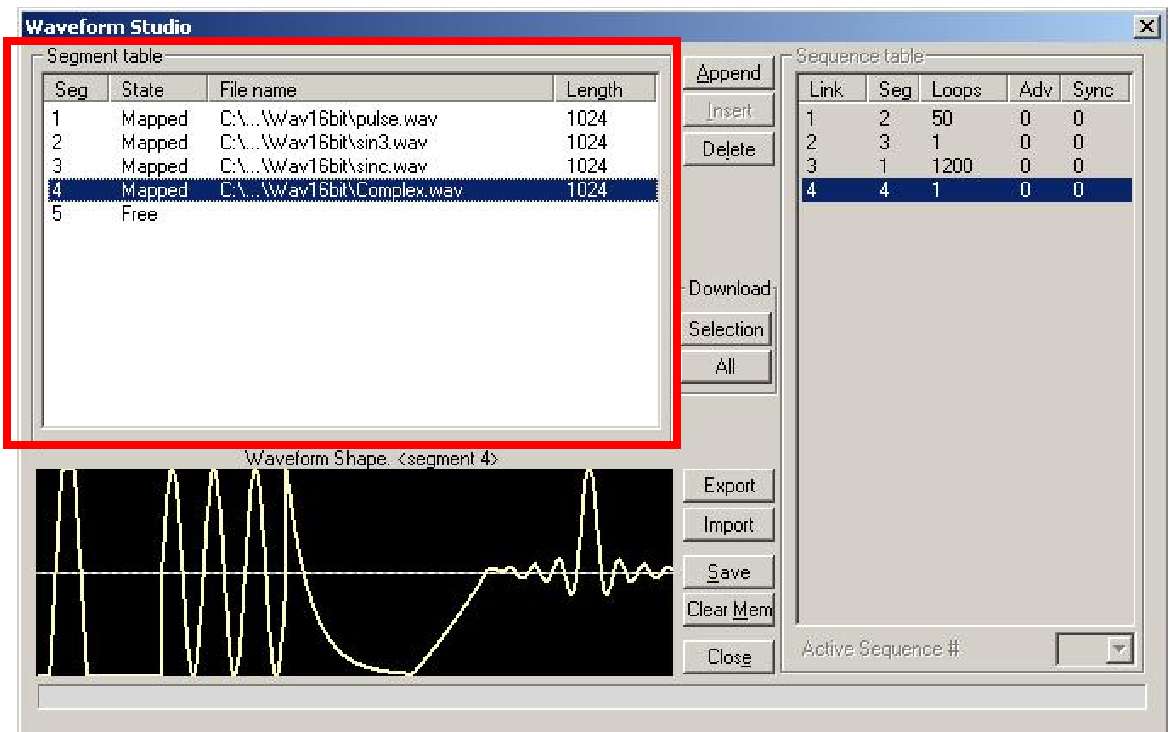

Figure 4-10, Waveform Studio

TIP

Point and click on one of the segments to show its shape in the Waveform Shape window.

Description of the various buttons in the Segment Table is given below.

Append – adds segment number at the end of the table

Insert – adds a segment above a highlighted segment line

Delete – removes a highlighted segment

(Download) Selection – downloads a highlighted segment only to the 3172 memory

(Download) All – downloads the complete table to the 3172 memory

Export – This allows exportation of Waveform Studio settings to another session

Import – This allows importation of Waveform Studio settings from another session

Save – saves current table settings

Clear Mem – wipes out the entire memory and clears the table for fresh settings

Close – removes the Waveform Studio from the screen. If you have not saved your work, the table setting will be lost.

Sequence Table

As was explained in the above, the waveform memory can be divided into smaller segments and up to 16 k segments can be defined and used as individual arbitrary waveforms. Having a limited size of waveform memory can, for some applications, pose a limitation however, if sections of the waveform are repetitive, one may use the sequence generator to take these segments and replay them as part of the complete waveform without losing valuable memory space and without scarifying waveform coherence and integrity. The tool for combining repetitive and multiple segments in one long waveform is called Sequence Generator and ArbConnection has a special dialog box where sequences are designed. This tool is called – Sequence Table.

Using the Sequence Table you can use waveforms that you already downloaded to the 3172 from the Segment Table, link and loop in random order to create one long and complex waveform that combines the individual memory segments.

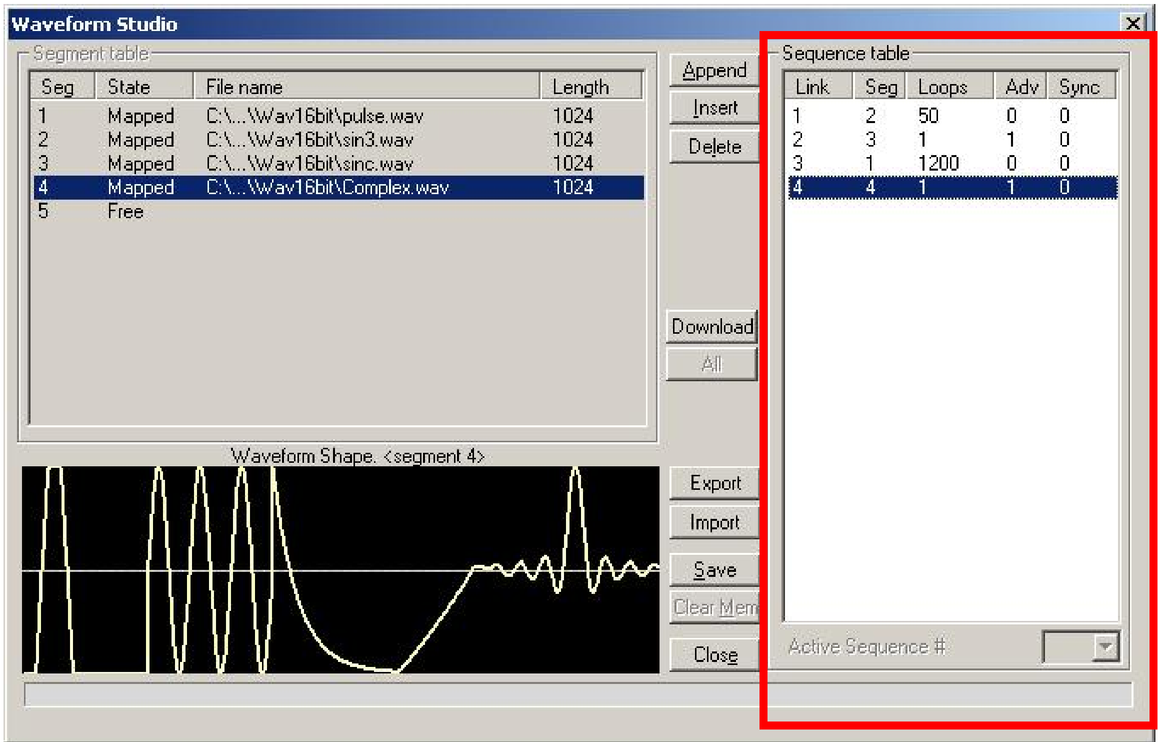

Figure 4-11, Sequence Table Example

The Sequence Table is highlighted in Figure 4-11. To access the Sequence table, click anywhere on the Sequence Table area. If it was not yet, it will turn white as opposed to the Segment Table area that turns gray.

There are five major elements that you should consider while programming a sequence table. They are: Link, Seg, Loops, Adv and Sync. These terms are explained below.

Link - This parameter defines an index array for the sequence generator. When generating sequences, the instrument steps though the links in descending order therefore, make sure that you enter your waveform segments in exactly the order you would like them at the output.

Seg - This parameter associates waveform segments with links. You can use different segments for different links or you can use the same segment for a number of links. There are no limitations to how you associate links to segments except that you cannot program assign segments to the sequence table that were not defined previously.

Loops – This parameter define how many times the segment will loop for the selected link. For example, if you program 2, the waveform will cycle twice through the same segment before transitioning to the next link.

Adv – This parameter flags the advance mode for a given segment. This flag is active when the advance mode is Stepped. When set to 0, the sequence will advance through the list automatically until a segment that is flagged 1 is encountered. When 1 is encountered, the generator will idle on this segment until an external trigger is applied. Learn more about the sequence advance modes in Chapter 3.

Figure 4-11 shows an example of a 4-step sequence of which the first waveform is made of segment 2, which will loop 50 times; segment 3, looping once; segment 1, looping 1200 times and segment 4, looping once. The Adv bits on links 2 and 4 are set to 1 and therefore, external triggers are required for the sequencer to step through these links.

Sync – This parameter flags if a bit marker will occur on the selected segment. Normal sync output is LCOM for the sequence mode however, if you want to use shortened and/or multiple sync pulses, change the sync output selection in the Arbitrary/Sequence Panel to BIT and the output will generate a pulse every time the sequence steps through a segment that has been flagged.

The control buttons on the left of the Sequence Table have the same functionality as for the Segment Table.

Use the **Append** key to add a step at the end of the sequence list. Use the **Insert** key to insert a step at the cursor location. The **Delete** key is used for deleting a step at the cursor position.

Click on the **Close** to discard of the contents of the dialog box without saving your last actions and to remove the sequence Table from the screen but click on the **Save** key if you want just to save your work before you close the dialog box.

The **Download** key has double action, it will download the sequence table to the instrument and will save the contents of your table so the next time you open this table, it will have the same contents as you saved in your previous session.

Half Cycle

The Half Cycle panel contains controls that select the half cycle functions and adjust the half cycle parameters. The half cycle functions are generated with variable and controllable delay between the halves. If triggered mode, one half at a time is generated as a result of a trigger signal regardless of the programmed delay value. The half cycle functions have different limitations compared to the standard functions; these are listed in Appendix A. The half cycle panel and the various parameters that control these functions are described below.

Figure 4-12, Half Cycle Panel

State

The State button turns on and off the half cycle waveforms function. The half cycle function can also be selected from the Output panel.

Shape

The Shape group has controls that select the shape of the half cycle function.

Parameters

The Parameters group has controls for programming the amplitude, offset, start phase and duty cycle. Each channel can have an independent set of these parameters.

The Modulation Panels

The Modulation functions were designed over seven separate panels, as shown in Figures 4-13 through 4-18. The panels are invoked by pressing the Modulation header and then one of the modulation panels that appear below it. These panels provide access to all modulation functions and their respective parameters. The modulation functions that are available on these panels are: FM (frequency modulation), AM (amplitude modulation), Sweep, ASK (amplitude shift keying), FSK (frequency shift keying) and PSK (phase shift keying) and Amplitude and Frequency Hopping.

When modulation run modes other than continuous are selected, there are two options that control the idle state between triggers: 1) Carrier baseline and 2) DC baseline. When the first option is selected, the instrument generates the unmodulated carrier frequency (CW) until a valid signal is applied. When the second option is selected, the instrument generates a DC level signal until stimulated to generate a modulation cycle. The modulation options, their associated parameters, and the various run mode options are described separately for each of the panels below.

Figure 4-13, Modulation Panels

FM

The FM panel (Figure 4-14) contains parameters for controlling the amplitude modulation function. To turn the FM function on and off, click on the FM button in the State group. The various groups in the FM panel are described below.

State
The State button turns on and off the FM function.

FM Parameters
This group contains parameters that allow complete control over the FM function. These are:

CW Frequency – The CW Frequency is the frequency of the pre-modulation carrier waveform. In case the modulating waveform is one of the built-in standard waveforms, the modulation will be symmetrical about the CW frequency setting.

Baseline – The Baseline parameter affects the output characteristics in one of the interrupted run modes (i.e., triggered, burst). In this case this parameter defines where the signal idles between triggers. There are two options: CW and DC. The DC option will set the idle state to a DC level, meaning that in between triggers, the output resides on a DC level and generates modulation when a trigger is accepted. The CW is similar except the signal idles on the pre-trigger CW frequency setting, executes the modulation upon receipt of a legal trigger signal and returns to continuous CW frequency output.

Figure 4-14, FM Panel

Standard FM Parameters - These parameters are active only when one of the built-in waveforms is selected as the modulating signal. These are: Sine, Triangle, Square, or Ramp. The modulation frequency, deviation and marker frequency control the standard FM modulation scheme.

Modulating Wave - Defines the shape of the modulating waveform. There are two basic options: Standard (built-in) waveforms and Arbitrary waveforms. If you do not need exotic waveforms, you can use one of the built-in standard wave shapes: Sine, Triangle, Square, or Ramp. These waveforms can be adjusted for their frequency and deviation range. On the other hand, you can select the arbitrary modulating wave option where you can use any shape, although you must load the modulating waveform from an external application, such as the FM composer in ArbConnection. Information on the standard and arbitrary FM functions is given in Chapter 3. Click on the button next to the required modulating waveform shape to select it.

Arbitrary FM Parameters – Allow adjustment of the sample clock of the modulating waveform. These parameters are active only when arbitrary modulating waveform option is selected. The modulating waveform must be downloaded from an external utility such as ArbConnection and the sample clock is programmed from this location.

To access the required parameter, click on the parameter's name and observe that the LED next to the required parameter turns on. The value that is associated with the lit LED is displayed on the digital display. You can use the dial, keyboard, or the [↑] [↓} keys to adjust the readout to the required setting. After you modify the reading, click on the Modify/Execute knob to update the 3172 with the new setting.

AM

The AM panel (Figure 4-15) contains parameters for controlling the amplitude modulation function. To turn the AM function on and off, click on the AM button in the State group. The various groups in the AM panel are described below.

State
The State button turns on and off the AM function.

AM Parameters
This group contains parameters that allow complete control over the AM function. These are:

CW Frequency –The CW Frequency is the frequency of the carrier waveform.

Baseline – The Baseline parameter affects the output characteristics in one of the interrupted run modes (i.e., triggered, burst). In this case this parameter defines where the signal idles between triggers. There are two options: CW and DC. The DC option will set the idle state to a DC level, meaning that in between triggers, the output resides on a DC level and generates modulation when a trigger is accepted. The CW is similar except the signal idles on the pre-trigger CW frequency setting, executes the modulation upon receipt of a legal trigger signal and returns to continuous CW frequency output.

Figure 4-15, AM Panel

Modulating Wave – Defines the shape of the modulating waveform. There are four built-in standard wave shapes: Sine, Triangle, Square, or Ramp. These waveforms can be adjusted for their frequency and deviation range. Click on the button next to the required modulating waveform shape to select it. The modulating waveform can be selected independently for each channel

Freq – Programs the frequency of the modulating waveform. Note that the frequency setting must be smaller than the CW frequency for the AM function to operate correctly. Note that the modulating frequency setting is common to both channels.

Depth – The Depth parameter programs the modulation depth, or index in percent of the un-modulated CW amplitude. The depth is symmetrical about the center of the CW amplitude. Each channel can have a unique setting of the modulation depth.

To access the required parameter, click on the parameter name and observe that the LED next to the required parameter turns on. The value that is associated with the lit LED is displayed on the digital display. You can use the dial, keyboard, or the [↑] [↓} keys to adjust the readout to the required setting. After you modify the reading, click on the Modify/Execute knob to update the 3172 with the new setting.

Sweep

The Sweep Modulation panel (Figure 4-16) contains parameters for controlling the sweep function. To turn the sweep function on and off, click on the Sweep button in the State group. The various groups in the sweep panel are described below.

State
The State button turns on and off the Sweep function.

Sweep Parameters
This group contains parameters that allow complete control over the sweep function. These are:

Baseline – The Baseline parameter affects the output characteristics in one of the interrupted run modes (i.e., triggered, burst). In this case this parameter defines where the signal idles between triggers. There are two options: CW and DC. The DC option will set the idle state to a DC level, meaning that in between triggers, the output resides on a DC level and generates modulation when a trigger is accepted. The CW is similar except the signal idles on the pre-trigger CW frequency setting, executes the modulation upon receipt of a legal trigger signal and returns to continuous CW frequency output. Note that in sweep modulation, the Start parameter replaces the CW value.

Function – The Function buttons select which of the waveforms will be swept. The sine wave is the default waveform and it is swept using the DDS circuit. The other two waveforms, triangle and square, are computed and the swept coordinates placed in the arbitrary memory. The calculation of the sine and triangle waveforms takes a long time (any time between seconds to minutes, depending on the complexity of the sweep) so should only be used when absolutely necessary in the application.

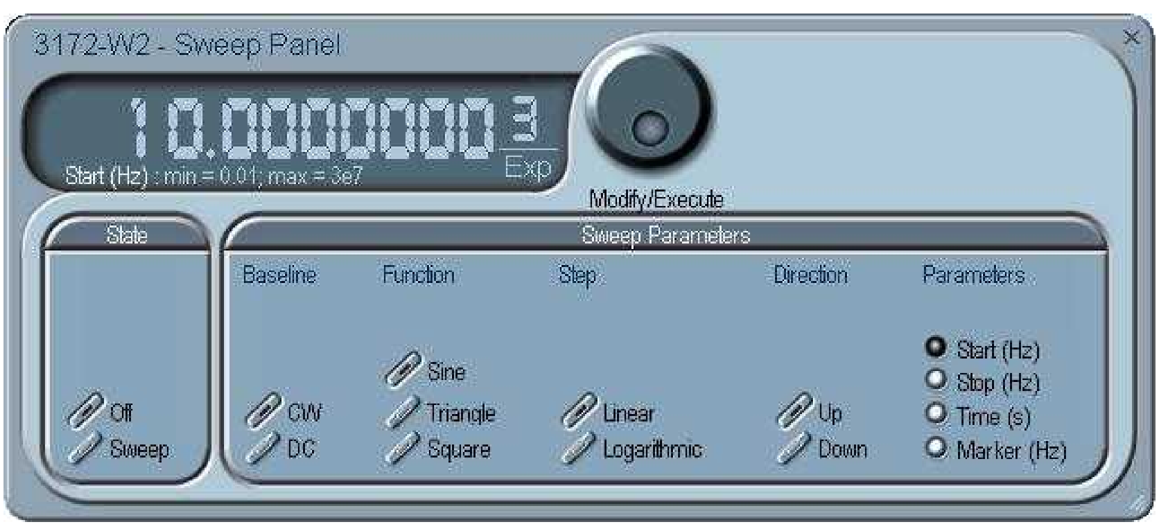

Figure 4-16, Sweep Modulation Panel

Step
Use these keys to select sweep step from two increment options: linear, or logarithmic.

Direction
Use these keys to program the sweep direction. Up selects a sweep from the Start to Stop sample clock setting and Down selects sweep from the Stop to Start sample clock setting. Refer to Chapter 3 of this manual to learn more about sweep operation.

Parameters
These allow the adjustment of the Sweep Start (CW), Stop and Sweep Time. You can also place a marker at a position programmed by the Marker parameter. To access the required parameter, click on the parameter's name and observe that the LED next to the required parameter turns on. The value that is associated with the lit LED is displayed on the digital display. You can use the dial, keyboard, or the [↑] [↓] keys to adjust the readout to the required setting. After you modify the reading, click on the Modify/Execute knob to update the 3172 with the new setting.

FSK/PSK/ASK

The FSK/PSK/ASK panel (Figure 4-17) contains parameters for controlling the shift keying modulation functions. To turn one of the functions on and off, click on the appropriate button in the State group. The various groups in this panel are described below.

State
The State buttons enable or disable the shift keying functions.

General
The General group contains parameters that are common to all of the shift keying functions. These are CW frequency and baseline.

CW Frequency – The CW Frequency is the frequency of the pre-modulation carrier waveform.

Baseline – The Baseline parameter affects the output characteristics in one of the interrupted run modes (i.e., triggered, burst). In this case this parameter defines where the signal idles between triggers. There are two options: CW and DC. The DC option will set the idle state to a DC level, meaning that in between triggers, the output resides on a DC level and generates modulation when a trigger is accepted. The CW is similar except the signal idles on the pre-trigger CW frequency setting, executes the modulation upon receipt of a legal trigger signal and returns to continuous CW frequency output.

FSK
The FSK group contains parameters that control the frequency shift keying function. These parameters are: control data, "0" and "1" frequencies, baud rate, and marker position.

Control Data – The Control Data button in the FSK group provides access to the data string that controls the sequence of base frequency and shifted frequency. It contains the list of “0” and “1” values that the output will repeatedly follow for its frequency shift keying sequence advance as programmed.

Figure 4-17, FSK/PSK/ASK Modulation Panel

“0/1” Frequency – In FSK, the carrier waveform (CW) has two frequencies: an initial frequency level which is set by the “0” frequency parameter and shifted frequency which is set by the “1” frequency. The control data table has a list of “0” and “1” values that flag when the frequency shifts from base to shifted frequency.

Baud – The baud parameter sets the rate at which the generator steps through the sequence of the FSK Control Data bits.

Marker Index – The marker index programs a step in the control data string to output a pulse at the SYNC output connector. The SYNC State button must be turned on to generate the FSK marker output.

PSK

The PSK group contains parameters that control the phase shift keying function. These are: control data, “0” and “1” frequencies, baud rate, and marker position.

Control Data – The Control Data button in the PSK group provides access to the data string that controls the sequence of base phase and shifted phase. It contains the list of “0” and “1” values that the output will repeatedly follow for its phase shift keying sequence advance as programmed.

“0/1” Phase – In PSK, the carrier waveform (CW) has two phase settings: an initial phase which is set by the “0” Phase parameter and shifted phase which is set by the “1” Phase. The control data table has a list of “0” and “1” values that flag when the phase shifts from base to shifted phase.

Baud – The baud parameter sets the rate at which the generator steps through the sequence of the PSK Control Data bits.

Marker Index – The marker index programs a step in the control data string to output a pulse at the SYNC output connector. The SYNC State button must be turned on to generate the PSK marker output.

To access the required parameter, click on the button below parameters sub-group until the LED next to the required parameter turns on. The value that is associated with the lit LED is displayed on the digital display. You can use the dial, keyboard, or the [↑] [↓} keys to adjust the readout to the required setting. After you modify the reading, click on the Modify/Execute knob to update the 3172 with the new reading.

ASK

The ASK group contains parameters that control the amplitude shift keying function. These are: control data, non-modulated and shifted phases, and baud and marker position.

Control Data – The Control Data button in the ASK group provides access to the data string that controls the sequence of base amplitude and shifted amplitude. It contains the list of “0” and “1” values that the output will repeatedly follow for its amplitude shift keying sequence advance as programmed.

“0/1” Amplitude – In ASK, the carrier waveform (CW) has two amplitudes: an initial amplitude level which is set by the “0” Amplitude parameter and shifted amplitude which is set by the “1” Amplitude. The control data table has a list of “0” and “1” values that flag when the amplitude shifts from base to shifted amplitudes.

Baud – The baud parameter sets the rate at which the generator steps through the sequence of the ASK Control Data bits.

Marker Index – The marker setting programs a step (index) in the control data string to output a pulse at the SYNC output connector. The SYNC State button must be turned on to generate the ASK marker output.

Amp/Freq Hop

The Amp/Freq Hop panel (Figure 4-18), contains parameters for controlling the hop modulation function. To turn one of the functions on and off, click on the appropriate button in the State group. The output has two hop options: Fixed and Variable. In Fixed mode, the output steps through the pre-assigned hop values at a constant rate, as programmed using the dwell time parameter. In Variable mode, the output dwells on each step for a period of time that is programmed in the Dwell Time field in the hop data table that is programmed when using the Variable Hold option.

The various groups in this panel are described below.

State
The State buttons enable or disable the hop functions.

General
The General group contains parameters that are common to all of the hop functions. These are CW frequency and baseline.

CW Frequency – The CW Frequency is the frequency of the pre-modulation carrier waveform.

Baseline – The Baseline parameter affects the output characteristics in one of the interrupted run modes (i.e., triggered, burst). In this case this parameter defines where the signal idles between triggers. There are two options: CW and DC. The DC option will set the idle state to a DC level, meaning that in between triggers, the output resides on a DC level and generates modulation when a trigger is accepted. The CW is similar except the signal idles on the pre-trigger CW frequency setting, executes the modulation upon receipt of a legal trigger signal and returns to continuous CW frequency output.

Amplitude Hop
The Amplitude Hop group contains parameters that control the amplitude hop function. These are: hop data, dwell control, dwell time and marker position.

Hop Data – The Hop Data button in the Ampl Hop group provides access to the data string that controls the sequence of amplitude hops. The hop data table contains a list of amplitude levels that the output steps through the amplitude levels of as programmed in the hop data table.

Fixed Hold – The hold parameter determines how long each step of amplitude dwells on this setting before it will step to the next amplitude setting. By selecting Fixed Hold, the hold time remains constant throughout the entire hop table.

Figure 4-18, Amp/Freq Hop Panel

Variable Hold – The Variable Hold parameter determines how long each step of amplitude dwells before stepping to the next amplitude setting. By selecting Variable Hold, the hold time changes automatically from one step to the next depending on the hold time value that is affixed to the hop step. The values can be programmed in the HOP Data table.

Dwell Time –The Dwell Time parameter programs the period of time that must elapse before the output amplitude hops to the next amplitude setting. Dwell Time is associated with the Fixed Dwell option only.

Marker Index – The marker index programs a step in the hop data string to output a pulse at the SYNC output connector. The SYNC State button must be turned on to generate the hop marker output.

Frequency Hop

The Frequency Hop group contains parameters that control the frequency hop function. These are: hop data, dwell control, dwell time and marker position.

Hop Data – The hop data button in the frequency hop group provides access to the data string that controls the sequence of frequency hops. The hop data table contains a list of frequencies and the output will step from one frequency level to another in the same order as programmed in the hop data table.

To access the required parameter, click on the button below the parameters sub-group until the LED next to the required parameter turns on. The value that is associated with the lit LED is displayed on the digital display. You can use the dial, keyboard, or the [↑] [↓} keys to adjust the readout to the required setting. After you modify the reading, click on the Modify/Execute knob to update the 3172 with the new setting.

Auxiliary Panels

The Auxiliary tab provides access to a group of panels that control some auxiliary functions (Figure 4-19).

There are three panels in this group: Counter/Timer, which provides access to the auxiliary Counter/Timer function; Pulse Generator, which provides access to the digital pulse generator function; and X-Instrument Sync for multi instrument synchronization control.

Figure 4-19, Auxiliary Panels

Counter/Timer

The Counter/Timer panel (Figure 4-20) contains controls that select the measurement function and adjust the counter/timer parameters for measuring external signals. The counter/timer measures signals that are connected to the TRIG IN input. The various parameters that control the counter/timer features are described below.

State

The State Group has controls to enable or disable the counter. And to reset the counter and arm it for the next measurement cycle. Note that when the counter function is turned on, all other waveform generation features of the 3172 are purged.

Measurement Function

The measurement function group has control to select the measurement function for the counter/timer operation. The 3172 can measure the following function: Frequency, Period, Period Averaged, Pulse Width, and Totalize. The totalize function has two options. If Totalize Infinite function is selected the input will count every legal pulse at the counter input, for an indefinite period of time, and displays the total number of pulses until the counter has been reset. If Totalize Gated function is selected, the input will count every legal pulse at the trigger input for a period of time that is defined with the Gate Time parameter.

Figure 4-20, Counter/Timer Panel

Display

The Display Group has controls to select the display mode and to select if the display shows measurement or gate time readings.

In normal mode, the counter is armed to receive signal at the trigger input. When signal is sensed, the gate to the counter opens for duration as programmed with the Gate Time parameter, processes the result, displays the reading and continues with the same process as long as the signal is available at the input.

In hold mode, the counter is armed to receive signal at the trigger input. When signal is sensed, the gate to the counter opens for duration as programmed with the Gate Time parameter processes the result, displays and holds the reading until the next Reset/Arm command.

To display and modify the gate time parameter, click on the Gate Time LED and modify the gate time per your requirements. Gate time rage is from 100 μs to 1 s. Normal counter/timer readings are displayed when the Reading LED is selected.

X-Instrument Sync

The X-Instruments Synchronization table provides a fast and easy method of synchronizing modules that reside within a single VXI mainframe. ArbConnection finds 3172s which may be synchronized and allows the selection of groups and synchronization paths and allows the setting of phase offsets between modules.

Figure 4-21 shows a list of modules that were detected by ArbConnection and listed in the Instruments Pool. This list can now be manipulated to form one or more groups of synchronized instruments. Use the procedure below to set up groups and to activate the synchronization.

First, notice the variety of instruments that are listed in the Instruments Pool. Actually, they all are the same 3172 units except they are mounted on different platforms for various applications. The 3172-W6 is comprised of a single arb the 3172-W6W6 has two arbitrary waveform generators embedded in the same module. The 3172-W6P2 is the standard, single-slot instrument that has the ability to replace a 3171 module in legacy systems. Information on the various 3172 configurations is given in Chapter 1.

Information on how to set up synchronization groups along with a description of the various buttons that control the multi-instruments synchronization function is given below.

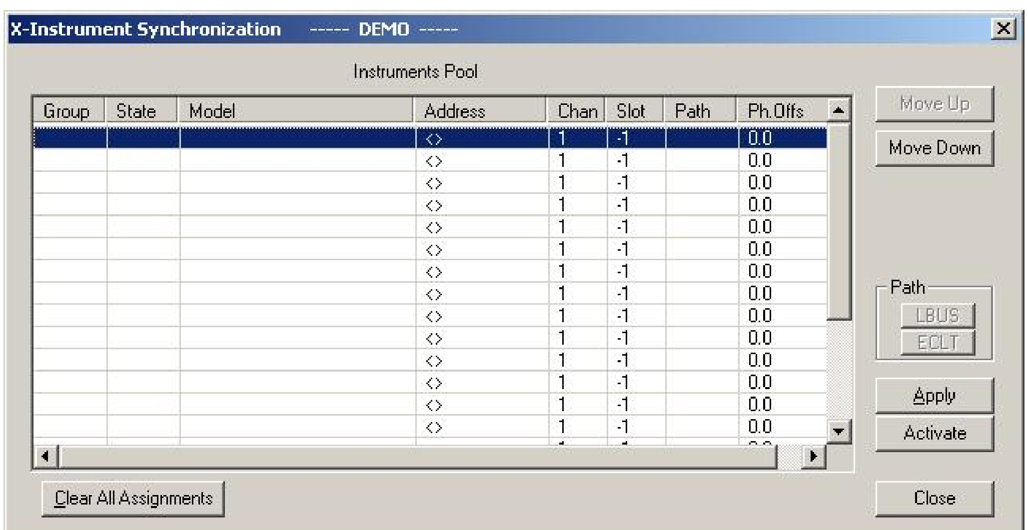

Figure 4-21, X-Instrument Synchronization Pool List

Group – is an edit field which is used for grouping one or more instruments into a set of instruments that share synchronization properties.

State – identifies the master or servant property of an instrument. Note that the first instrument in the group list is always set automatically as the master. If you want it as servant, you can use the Move Up and Move Down buttons to move the module higher or lower in the synchronization hierarchy.

Model – shows all instruments from the 3172 family that ArbConnection detected in the chassis. Leave the group field blank if you do not wish to synchronize a particular module.

Address – shows the logical address associated with the listed module. The address is set using a DIP switch which can be accessed when the module is removed from the chassis.

Chan (Channel) – this field is relevant for the 3172-W6W6 model which has two arb units installed in a single VXI slot. In this case, each instrument operates as a stand-alone generator but also can be configured as a dual-channel instrument. The example in Figure 4-22 shows this instrument with address 13 installed in slot 3. Channel 1 is always the top instrument and channel 2 is installed below and if selected as a group, by default, channel 1 becomes the master module. There is no way to exchange channel designation but if you move channel 2 up, its state is re-configured to master and channel 1 to servant.

Slot – shows the slot number where ArbConnection found a specific module. The location of the module is extremely important because it defines how it may be synchronized, as explained in the Path description below.

Path – defines the connection between synchronized modules. This field has three options: ADJ, LBUS and ECLT.

> **ADJ** – defines a connection between two adjacent W6 modules in a single 3172 carrier. Notice Figure 4-22; it shows the model 3172-W6W6 in slot 3. After grouping in group 1, the two instruments are automatically assigned the ADJ path. This cannot be changed because of the nature of synchronization of two instruments in a single slot. Also note that Channel 1 is now master and channel 2 is the servant.
>
> **LBUS** – defines a connection between adjacent slots using the VXI backplane local bus lines. In this case, the master is always on the left and the servant units are adjacent to the master on the right side. It is not allowed to break the chain of instruments because local bus lines connect instruments in a daisy-chained link. Note in Figure 4-23 that slots 7, 8 and 9 are daisy-chained through the local bus. The instrument in slot 7 is the master unit and the others are servants.
>
> **ECLT** – defines a connection between any slot using the VXI backplane ECLTrg lines. In this case, the master can be assigned to any instrument in the chassis regardless if it is on the left, right or mixed with other instruments in the chassis. The example in Figure 4-24 shows how to form two groups in a single chassis where group one synchronizes two channels and group 2 synchronizes three different instruments in a specific master-servant configuration. Note that slot 9 was moved up and assigned master while slots 7 and 8 became

servants.

Ph. Offs (Phase Offset) – defines an offset between the master module and its servants. Note that the master instrument can also be set with an offset but then the final offset between modules will be the difference between the offset settings of the salves to the master.

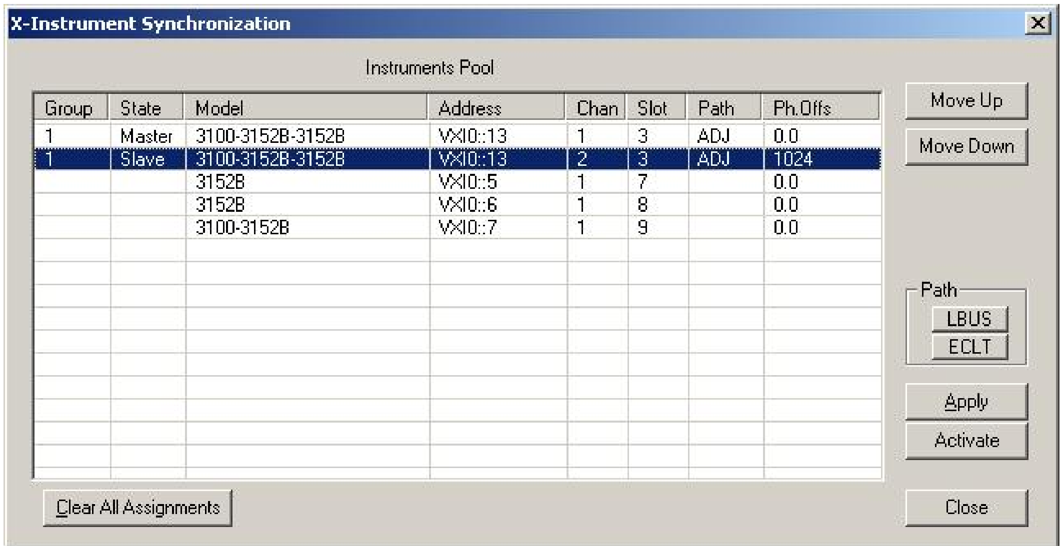

Figure 4-22, Adjacent Synchronization between Two Instruments

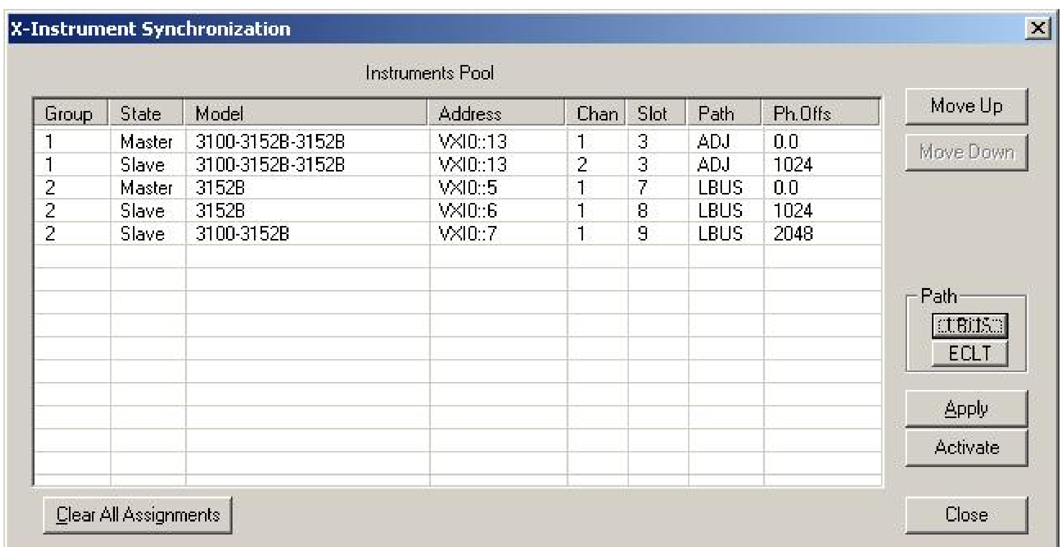

Figure 4-23, LBUS Synchronization between Adjacent Slots

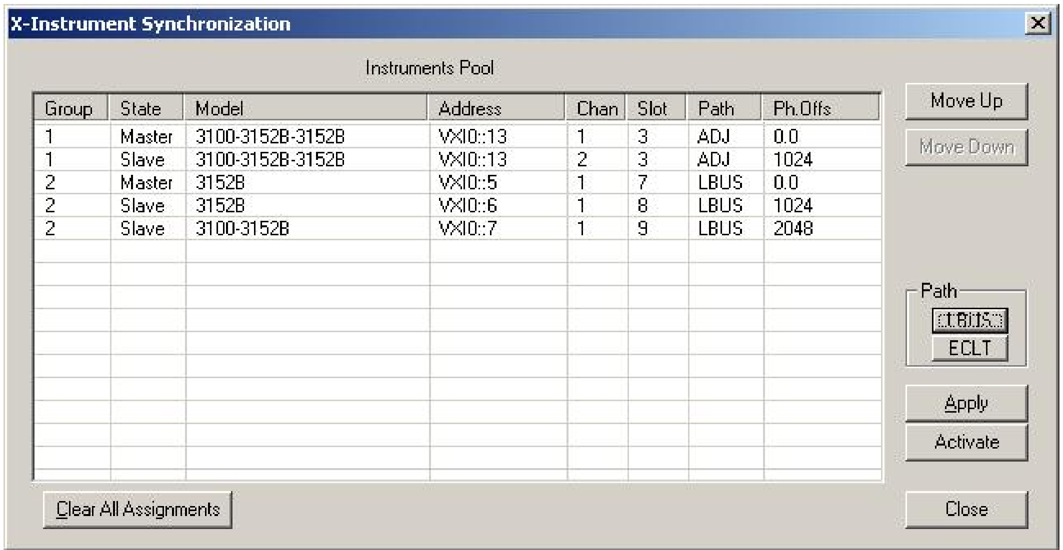

Figure 4-24, ECLT Synchronization Example

So far, the X-Instruments Synchronization fields were discussed and described. The following describes the functions of the buttons.

Clear All Assignments – used to completely reset the table. Note that only editable fields are affected by this action. Once pressed, the table will look as shown in Figure 4-21.

Move Up – used to change the position of a module to place it toward the top of the group and master status. Note that this operation affects the line that is currently highlighted.

Move Down – used to change the position of a module so it is placed below its current position in the group and can be used to demote a master to a servant. Note that this operation affects a line that is currently highlighted.

Path (LBUS/ECLT) – is used for selecting the connection path. LBUS specifies a VXI local bus connection and requires that the master is plugged into the leftmost position and all servants are plugged into adjacent slots to the right to the master module. ECLT specifies the VXI backplane ECLTrg line synchronization mode. For this mode, the location of the master and servant units in the chassis is not crucial because the trigger lines run across the backplane through all slots.

Apply – used to prepare the instruments for the synchronization sequence. This button must precede the activation of the synchronization process.

Activate –Once this button is pressed, the instruments will synchronize in the selected groups and the master will control the timing of the servant modules.

Close – terminates the current session but does not change the synchronization status.

The System Panels

The System tab (Figure 4-25) provides access to a group of panels that control some general system parameters and provides access to calibration. There are two panels in this group: General/System, which provides access to some system commands, utilities and filters; and Calibration, which provides access to the remote calibration utility. Note however, that access to the calibration panel is permitted to qualified service personnel and requires a user name and password. Information on how to access the calibration panel is given in Chapter 7.

Figure 4-25, System Panels

General/Filters

The General/Filters panel (Figure 4-26) provides access to some general system common commands, allows read back of information stored in the flash and provides a means to add filter(s) in series with the output path. The General/Filters panel and the various parameters that control its functions are described below.

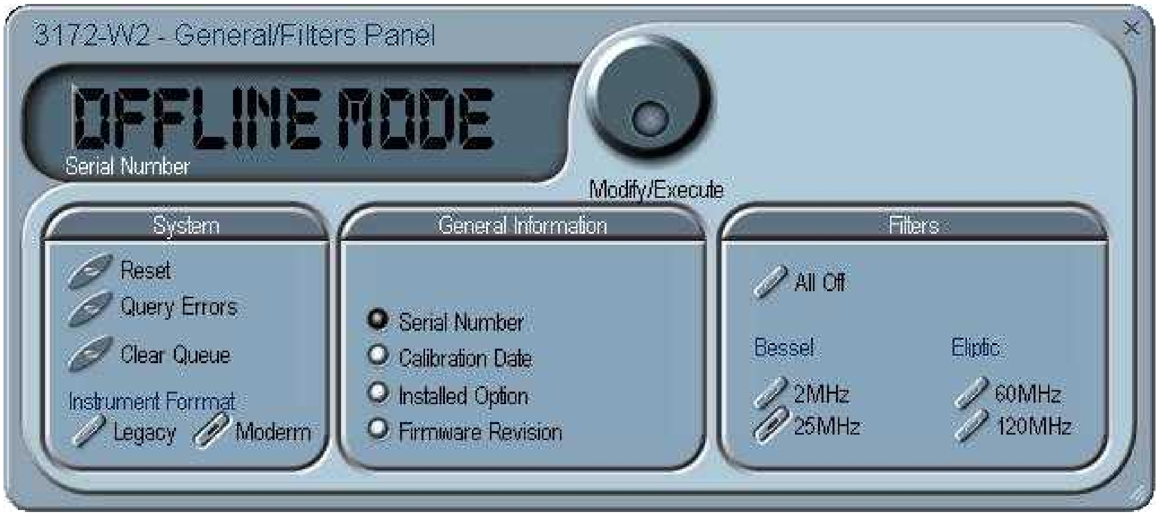

Figure 4-26, General/Filters Panel

System

The System group has three buttons that are normally associated with system control. These are:

Reset – generates a soft reset to the instrument controls and dialog boxes and modifies all parameters to factory default. A list of factory defaults is given in Chapter 5.

Query Error – queries the 3172 for programming errors. This command is normally not necessary because ArbConnection won’t generate settings conflicts or syntax errors. But, when sending SCPI commands to the instrument using the Command Editor, errors can be generated. An error query allows these errors to be monitored.

Clear Queue – clears the error queue. The error queue can buffer up to 35 errors and then generate a queue overflow message while ignoring new errors. This command clears the error queue and allows fresh errors to be captured.

General Information

This general information group of buttons is used for displaying or monitoring parameters stored in flash memory. These are: Instrument serial number, Last calibration data, 3172 installed options, and the installed firmware revision.

Filters

The Filters group has a set of selectors that select the filter characteristics. Filters can be turned on and off freely as long as you are not generating a standard sine waveform. The following filter options are available:

All Off – no filter is applied to the output path
2MHz – a Bessel type filter that has a 20 MHz cutoff frequency.
25MHz – a Bessel type filter that has a 25 MHz cutoff frequency.
60MHz – an Elliptic type filter that has a 60 MHz cutoff frequency.
120MHz – an Elliptic type filter that has a 120 MHz cutoff frequency.

Calibration

The Calibration panel (Figure 4-27) provides access to remote calibration. To access the remote calibration panel, you will need to have a valid User Name and Password. Proper training is required to perform calibration. Information on how to access the calibration panel and how to perform the calibration is provided in Chapter 7. The picture below is just for reference.

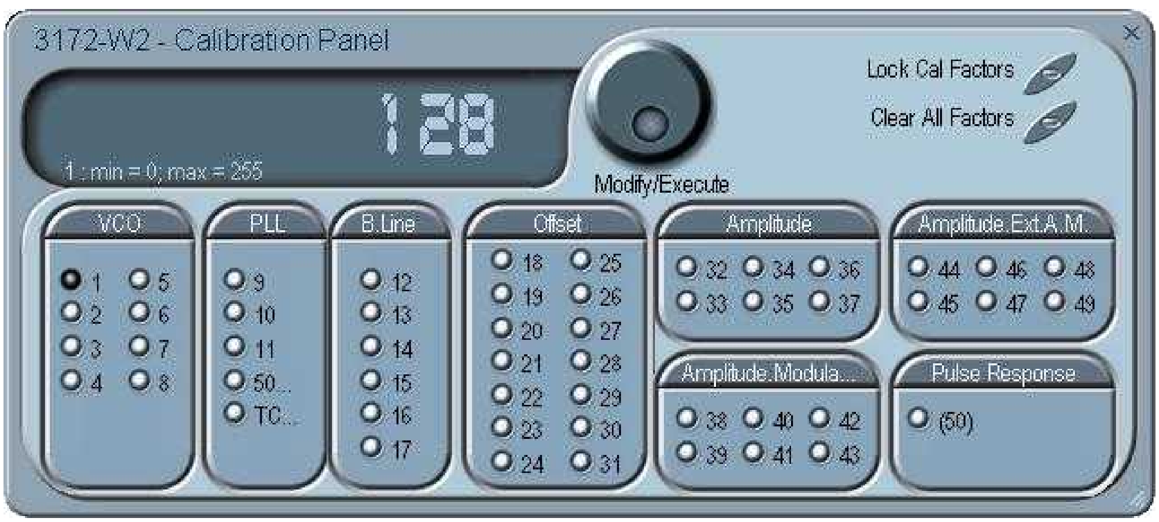

Figure 4-27, Calibration Panel

The Composers Panels

The Composers tab provides access to a group of composers that allow the generation and modification of arbitrary waveforms, pulse shapes, arbitrary frequency modulation, and 3D profiling.

There are four waveform composers built into ArbConnection:

Wave – for generating arbitrary waveforms. Arbitrary waveforms can be generated from standard libraries, from an equation editor, or imported into the composer from external utilities such as MatLAB^{TM}. The waveforms can be edited and stored to disk for future use.

Pulse – for generating complex pulse trains. Unlike a standard pulse generator, you can design and edit multiple pulse trains with linear transitions and variable amplitudes.

FM – for generating arbitrary frequency modulation profiles without being limited by the standard sine, triangle and square modulating shapes, and

3D – for generating chirps and simultaneous variations of amplitude, frequency and phase on each channel, separately.

Figure 4-28, Composers Panel

The Wave Composer

Because the 3172 is an arbitrary waveform generator, it has to be loaded with waveform data before it can start generating waveforms. The waveform generation and editing utility is part of ArbConnection and is called the Wave Composer. This program gives you tools for the definition of arbitrary waveforms. It can also convert waveform data from other products such as oscilloscopes, and use the data directly as waveforms. The program is loaded with many features and options so use the following paragraphs to learn how to create, edit, and download waveforms to the 3172 using the Wave Composer.

To launch the Wave Composer, point and click on the Wave tab in the Composers section of the Panels bar. Figure 4-29 shows an example of the wave composer. The Wave Composer has three main sections: the Menu bar, Toolbar, and Waveform graph. Refer to Figure 4-29 throughout the description of these sections.

The Wave Composer Menu Bar

The Wave Composer menu bar provides access to standard Windows operations such as File, Edit, and View. In addition, there are ArbConnection-specific operations such as Wave and System.

In general, clicking on one of the menus pulls down a list of commands. Clicking on a listed command may then either open a dialog box or generate an immediate action. For example, clicking on File and then Exit will cause the immediate termination of the Wave Composer. On the other hand, clicking on Wave and then on Sine, will open a Sine Wave dialog box that lets you program and edit sine wave parameters. The various commands in the menu bar are listed and described below.

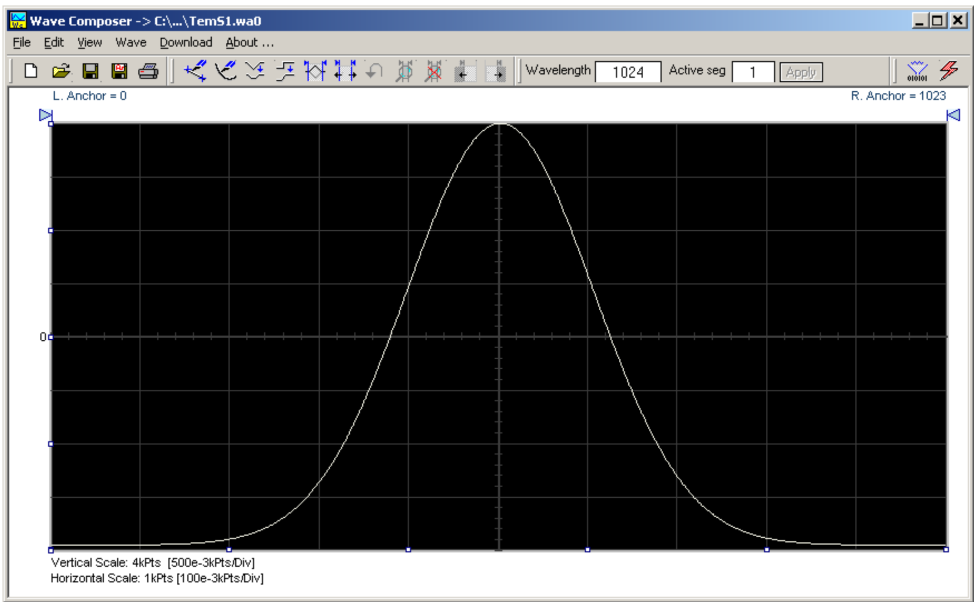

Figure 4-29, Wave Composer Opening Screen

File Menu

The File menu has four selections that control waveform file operations. This menu also can be used to print the active waveform or to exit from Wave Composer. Descriptions of the menu selections from the File pull-down menu are given below.

New Waveform

The New Waveform (Ctrl+N) menu item removes the current waveform from the graph window. Changes made to the waveform graph should be saved before using the New Waveform menu command because this function is destructive to the displayed waveform.

Open Waveform…

The Open Waveform… (Ctrl+O) menu item lets you choose a previously saved waveform file and load it to the waveform graph. This function can also import waveform files of various types to the Wave Composer. The Open Waveform… menu function can import ASCII, *.CSV (comma delimited text), *.PRN (space delimited text), *.0* (LeCroy binary) format, and others. The Open dialog box in Figure 4-31 shows the various file extensions that can be opened into the Wave Composer environment. The file that is opened is automatically converted to the binary *.wav format.

Save Waveform

The Save Waveform (Ctrl+S) menu item lets you store the active waveform as a binary file with a *.wav extension. If this is the first time you save your waveform, the Save Waveform As… command will be invoked automatically, letting you select the name, location, and format for the waveform file.

Save Waveform As…

Use the Save Waveform As… menu item the first time you save your waveform. It will let you select a name, location and format for your waveform file.

Print

Lets you print the active waveform graph The standard printer dialog box will appear and will let you setup the printer and print the waveform graph.

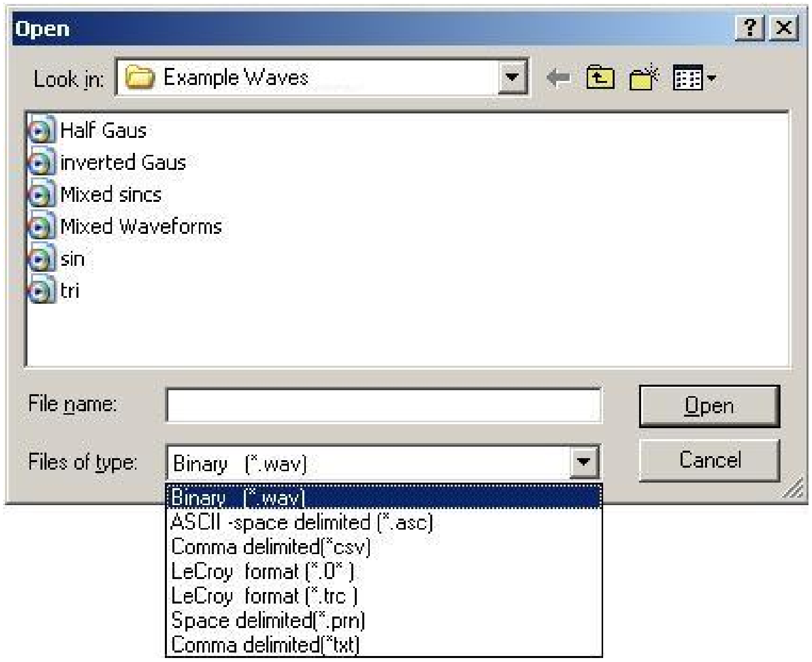

Figure 4-30, Open Waveform Dialog Box

Exit

Ends the current Wave Composer session and takes you back to the Panels screen. If you made changes to your waveform since it was last saved, Wave Composer will prompt you to Save or Abandon these changes.

Edit Menu

The Edit menu is used for manipulating the waveform that is drawn on the graph. The edit operations are explained below.

Autoline

Autoline mode lets you draw one or more connected line segments. To draw a line in Autoline mode, click the left mouse button at the start point. Click again at the next point to complete the line segment, repeating this way until finished creating connected line segments. Click on the right mouse button to terminate Autoline mode.

Sketch

Sketch mode lets you draw freehand segments. To draw in sketch mode, on the waveform graph, drag the cursor using the left mouse button. Release the mouse button when you want to stop. Use the right mouse button to terminate Sketch mode.

Filter

The Filter operation is calculated using a moving average. This is done by recalculating each point as an average of a number of symmetrical points adjacent to each point. You can filter the entire waveform, or you may choose to filter a segment of the waveform by placing the anchors as boundaries on the left and right of the segment.

Smooth

The Smooth operation lets you smooth out rough transitions in your waveform. This is done mathematically by multiplying the waveform by the nonlinear portion of a cubic curve.

The Smooth operation is done on segments of the waveform graph that are bounded by anchors. Anchor operations are described later in this chapter. Place the anchors to the left and right of the waveform segment to be smoothed and select the Smooth operation. The waveform will change its shape immediately to follow the mathematical pattern of a cubic curve.

Note that small segments with fast transitions, when combined with cubic functions, tend to generate even larger transitions. Therefore, be sure to omit such sections of the waveform when you use the Smooth operation.

Invert

The Invert operation lets you invert the entire waveform or a selected segment of a waveform. The waveform is inverted about the 0-point on the vertical axis.

Trim Left

The trim left command lets you trim waveforms to the left of the anchor point. This command is grayed out if the left anchor was not moved from its original left position. The waveform is trimmed and the point at the left anchor becomes the first point of the waveform.

Trim Right

The trim right command lets you trim waveforms to the right of the anchor point. This command is grayed out if the right anchor was not moved from its original right position. The waveform is trimmed and the point at the right anchor becomes the last point of the waveform.

Unmark
The Unmark operation removes the anchors from the waveform screen and resets anchor positions to point 0 and to the last waveform point.

Undo
The Undo command undoes the last editing operation.

View Commands

The View menu includes operations that let you view various sections of the waveform graph. View operations include: Zoom In, Zoom Out, Hide/Show Toolbars, and Channel selection (for dual channel units only). Descriptions of the view operations are given below.

Zoom In
The Zoom In operation operates between anchors. Anchors are shown as left pointing and right pointing triangles. The default position of the anchors is the start and the end of the waveform. To move an anchor to a new location, drag the anchor to the left or right as required. If you move the left anchor to the right and the right anchor to the left, the area between the anchors will fill the entire graph when the Zoom In operation is used.

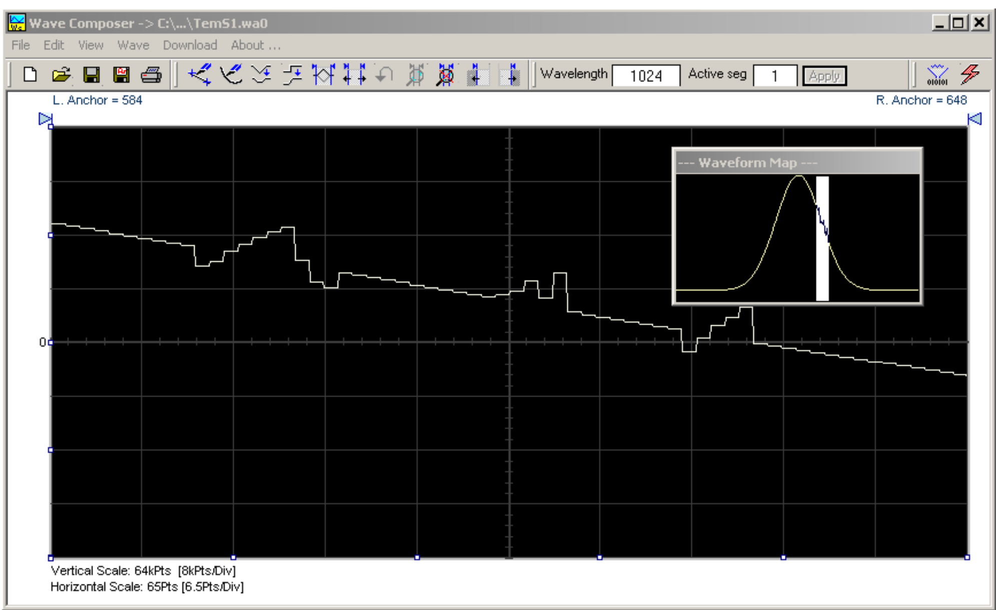

Figure 4-31, Zooming In on Waveform Segments

Looking at the Waveform Map in Figure 4-31, note that the white portion is the zoomed in area. Drag the white area with your cursor to peruse a zoomed in view of any portion of the waveform.

While zoomed in, you can invoke Autoline or Sketch mode.

Zoom Out
The zoom out operation restores the graph to display the complete

waveform.

Wave Menu

The Wave menu let you draw standard waveform functions on the graph. The Wave command has a library of 8 standard waveforms: Sine, Triangle, Square, Sinc, Gaussian, Exponent, Pulse, Noise, and DC. It lets you specify a cardiac ECG waveform or a Pulse Width Modulated (PWM) waveform. It also lets you create waveforms using the Equation Editor. Information on how to create waveforms using the Wave menu is given below.

Creating Waveforms From the Built-in Library

You can create any waveform from the built-in library using the Wave menu by clicking on one of the standard wave options to open a dialog box. An example of the Sine waveform dialog box is shown in Figure 4-32. This dialog box is similar to the rest of the waveforms, so the other waveform dialog boxes will not be described here.

Creating Sine Waveforms

Use the following procedure to create sine waveforms from the built-in library. Click on Wave, then sine… and the dialog box as shown in Figure 4-32 appears. You can now start programming parameters that are available in this box.

Start Point – Defines the first point where the created wave will start. Note that if you change the start point the left anchor will automatically adjust itself to the selected start point. The example shows start point set at point 0.

End Point – Defines where the created waveform will end. Note that as you change the end point the right anchor will automatically adjust itself to the selected end point. The example shows end point set at point 499.

Cycles – The Cycles parameter defines how many sine cycles will be created within the specified start and end points. The example below shows five sine cycles.

Amplitude – 16-bit resolution waveforms have 65,536 levels. The Amplitude parameter defines how many of these steps are used for generating the sine. The example is showing a sine waveform with maximum peak-to-peak amplitude for a 16-bit waveform. Any number range below the maximum and minimum values generates a sine with reduced dynamic range.

Start Phase – The start phase parameter defines the angle at which the sine will start. The example shows start phase of 90°.

Power – The example shows sine cubed. Sine to the power of 1 will generate a perfect sine. Power range is from 1 through 9.

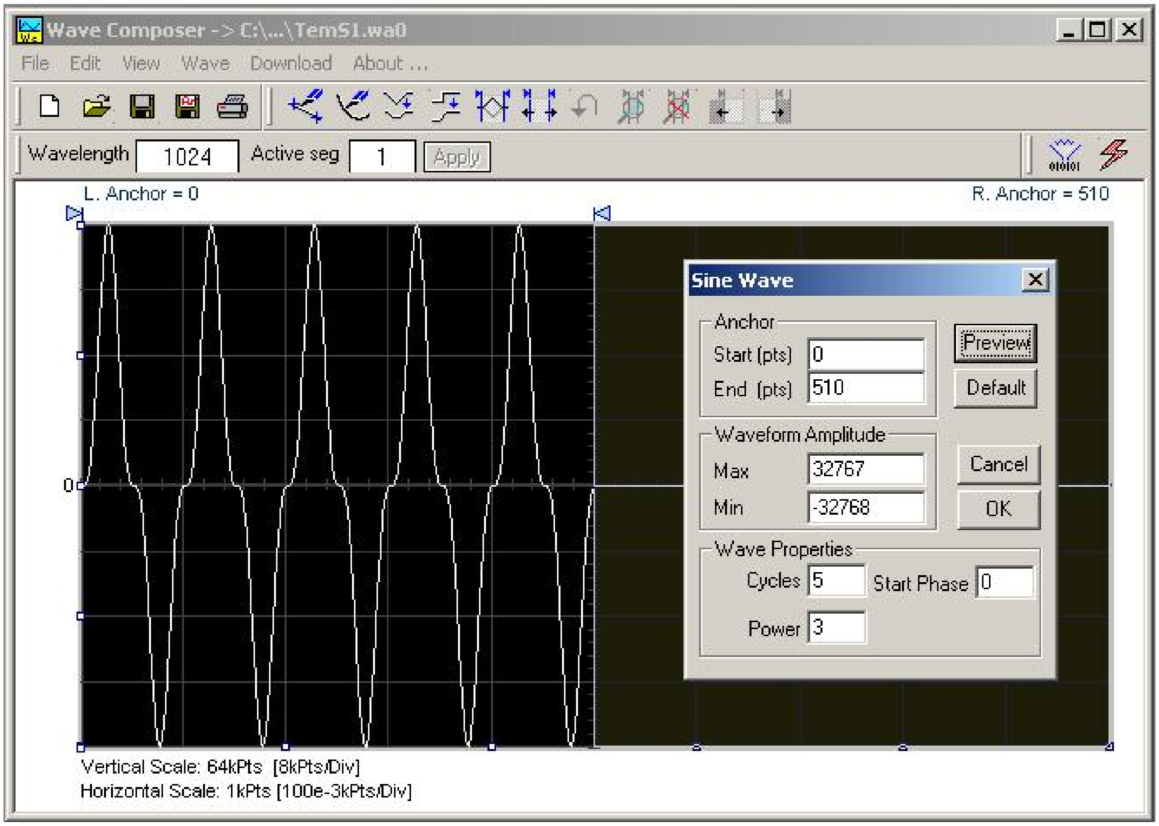

Figure 4-32, Generating Distorted Sine Waves from the Built-in Library

The Toolbar

The toolbar contains icons for editing the waveform screen, icons for saving and loading waveforms, fields for selecting an active channel and for adjusting segment length and more. The Toolbar is shown in Figure 4-33. For the individual icons, refer to the descriptions above of the Wave Composer Menus.

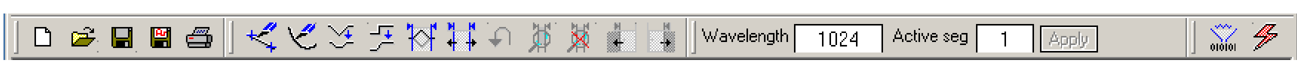

Figure 4-33, Toolbar Icons

The Waveform Screen

Waveforms are created and edited on the waveform screen. Figure 4-34 shows an example of a waveform created using the equation editor and the anchors to limit generation of the waveform to between points 100 and 900. The various elements of the waveform graph are described below.

The waveform graph has two axes – vertical and horizontal. Both axes are divided into points.

The vertical axis represents 16-bits (64k levels) of vertical resolution when modern mode operation is selected.

The horizontal axis, by default has 1024 points (from point 0 to 1023). This number can be changed using the Wave Length field in the Toolbar. The maximum length depends on the memory installed in your instrument. The wave composer will let you define the horizontal axis to a maximum of 1 Meg words with standard 1MB memory and 4 Meg words with the 4MB memory expansion option (where available).

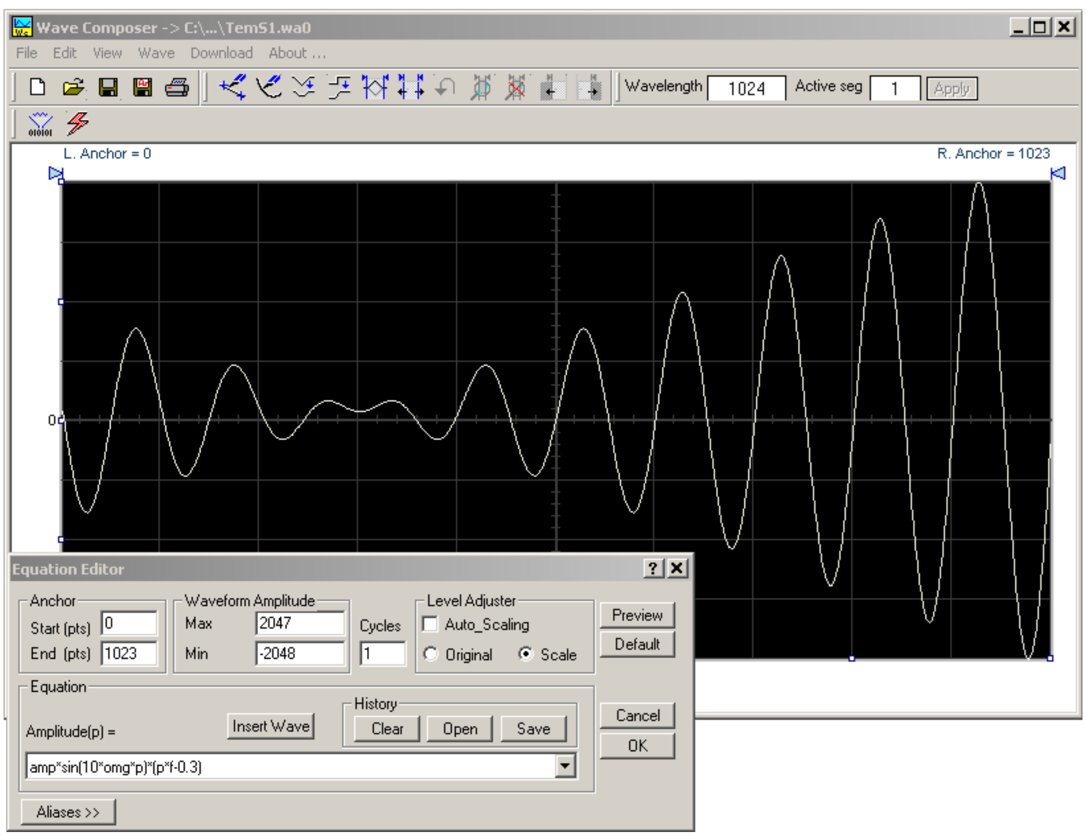

Figure 4-34, Waveform Screen

Notice that on the left top and right top there are two triangles pointing to the center of the screen. These are the anchors. The anchors are used as the start and end pointers for waveform creation. For example, if you want to create a sine waveform between point 100 and point 500, you place the left anchor at point 100 and the right at point 500 and then generate the sine from the built-in library.

There are two ways to control anchor placements.

1) Drag the left anchor triangle to the desired left position. Do the same for the right anchor. Notice that the anchor coordinates appear at the top of the waveform screen and that they change to correspond with your anchor placements.

2) You can also place your anchors in a more precise manner from the waveform library by programming the start and end points for the waveform. An example of anchor placement using the sine dialog box is shown in Figure 4-32.

Finally, when you are done creating and editing your waveform, you can save your work to a directory of your choice. The name of the waveform file will be displayed in Wave Composer's title bar, including the path.

Generating Waveforms Using Equation Editor

A more general purpose way to create waveforms using ArbConnection is to use Equation Editor. Equation Editor let you write equations for the desired waveform and lets ArbConnection calculate the values and display them on the graph. Equation Editor detects syntax errors and can auto-scale your waveforms so that no dynamic range is lost.

When you invoke Equation Editor, the dialog box shown in Figure 4-35 appears. The following paragraphs describe how to use the features of Equation Editor.

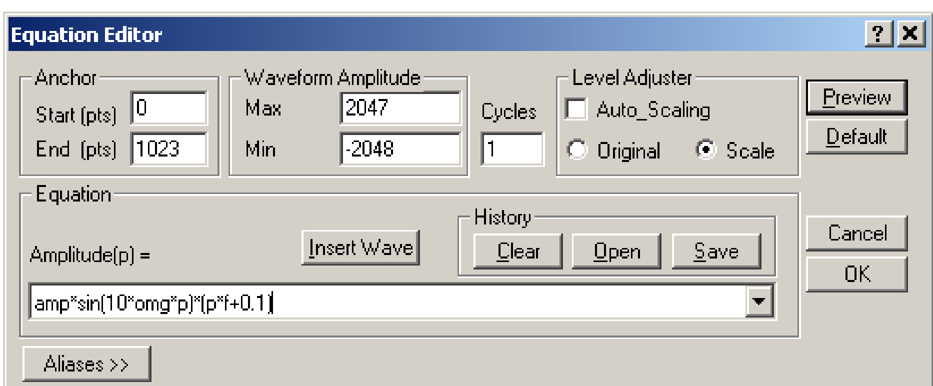

Figure 4-35, Equation Editor Dialog Box

There are four sub-group parameters in the equation editor plus control buttons and the equation field. These parts are described below.

Anchor

The anchors define start and end point at which the equation will be generated. By default the anchors are placed at the start and the end of the horizontal (time) scale however, the equation can be limited to a specific time scale by moving the anchor points from their default locations.

Start – defines the first point where the created wave will start. Note that if you change the start point the left anchor will automatically adjust itself to the selected start point.

End – defines where the created waveform will end. Note that as you change the end point the right anchor will automatically adjust itself to the selected end point.

Waveform Amplitude

The vertical axis of the Wave Composer represents 16-bits of vertical resolution. That means that the equation is computed, resolved and generated with 1/65,536 increments and accuracy. The Waveform Amplitude fields in the Equation Editor are used in two cases: 1) when the “amp” parameter is used in the equation or 2 if the Level Adjuster is set to Auto. Information on these two operations is given later.

Max – defines the positive peak of the vertical axis

Min – defines the negative peak of the vertical axis

Cycles

The Cycles parameter defines how many waveform cycles will be created within the specified start and end anchor points.

Level Adjuster

The Level Adjuster is a convenient tool that helps you adjust the amplitude and offset without modifying your equation. The Level Adjuster mode does not interfere with your calculations and displays the waveform as computed from your equation. The only difference is that your final calculations are stretched or shrunk or offset on the vertical scale to fit the new amplitude and offset boundaries.

If you change the Max and Min setting in the Waveform Amplitude fields and press the Adjust key, your waveform will offset immediately without changing the equation. The same way, you can also change amplitude only or both amplitude and offset. If you check the Manual option, you’ll have to click on the Adjust button for the Waveform Amplitude parameters to take effect. The Adjust button name will change to Restore and back to Adjust if you click on it again. If you check the Auto option, your waveform will be created automatically with the new Amplitude setting.

Equation

The Equation group has four buttons and the equation field. You will be using the Equation field for writing your equations. Equation syntax and conventions are discussed in the following paragraphs. The *Remove* button clears the equation field so you can start typing a new equation. Click on the *Store* button to store your equation if you intend to use it again. The Browse button provides access to waveform pre-stored files in your computer for combining them in new equations. The *Operands* button expands the bottom of the dialog box to show the operands you can use with your equation.

While you type and store equations, they are collected in a history file and can be used again by expanding the history log from the equation field.

Control Buttons

There are four control buttons at the right corner of the dialog box. Use the *Preview* button to preview an image of your equation, or use the *OK* button to place your waveform on the waveform screen and to leave the dialog box on the screen. The *Default* button restores the parameters in the equation editor to their original factory default values. The *Cancel* button will remove the dialog box from the screen and will discard of any waveforms that you previewed with your Equation Editor.

Writing Equations

The Equation Editor lets you process mathematical expressions and convert them into waveform coordinates. As you probably already know, waveforms are made of vertical samples. The number of samples on your waveform is determined by the wavelength parameter. For example, if you have 1024 horizontal points, your equation will be computed along 1024 points as a function of the vertical scale. Each vertical sample is computed separately and placed along the horizontal axis. The points are graphically connected to form a uniform and continuous waveform shape however, if you zoom in on a waveform line, you'll see that the points are connected like a staircase. In reality, the 3172 generates its waveforms exactly as shown on the screen but, if the waveform has many horizontal points, the steps get smaller and harder to see without magnification.

Equations are always computed as a function of the vertical (Amplitude) axis therefore the left side of your equation will always look as Amplitude(p) =, where “p” is the equation variables in units of waveform points. You can write equations with up to 256 characters. If the equation is too long to fit in the visible field, parts to the left or right will scroll off the ends.

Equation Conventions

Equations are written in conventional mathematical notation. You may only enter the right part of the equation. The only limitation is that the equation must be of a single variable that is directly related to the current horizontal axis setting. Case is not important and spaces are ignored. Numbers are entered in scientific notation. All calculations are done with double precision. For trigonometric functions, all angles are expressed in radians.

A number of constants are provided: e, which is the base of the natural logarithm; pi, which is the circumference of a unit-diameter circle; per, which equals the programmed horizontal range; f, which equals 1 /per; omg, which equals 2 * pi * f, and numbers in the range of -1E^20 to 1E^20.

There are three classes of precedence: ^ (raise to power) has the highest precedence; * (multiply) and / (divide) come second; + and - have the lowest precedence. Parentheses may be used to change the order of precedence. The following table summarizes the mathematical expressions and their respective abbreviated commands that can be used with Equation Editor.

Equation Editor Operands

^	Raise to the power
*	Multiply
/	Divide
+	Add
-	Subtract
( )	Parentheses
e	Base of natural Logarithm
pi (π)	Circumference of unit-diameter circle
per	Horizontal wavelength in points
f	I/per
omg (ω)	2*π*f
amp	Amplitude in units of points or Hertz
sin(x)	The sine of x*
cos(x)	The cosine of x*
tan(x)	The tangent of x*
ctn(x)	The cotangent of x*
log(x)	The base IO logarithm of x*
In(x)	The natural (base e) logarithm of x*
abs(x)	The absolute value of x*
-1E^20<>1E^20	Numerals, equation constants

*substitute your mathematical expression for x

After you get familiar with the operands and conventions, you can try a few simple equations and see how they create waveforms.

Typing Equations

Recall that a straight line is defined by Y as a function of X as in the equation Y = mX + b. You can use this to generate a straight line using Equation Editor. Assuming first that p = 0, try this:

Amplitude(p)=1000

Press [Preview] and see what you get. Of course, you get an uninteresting line that runs parallel to the X-axis. Now, let's give the line some angle by typing:

Amplitude(p)=-2*p+2000

Press [Preview] and see that the line slopes down. It may still be not very interesting however, pay close attention to the convention that is used in this equation. You cannot type: Amplitude(p)=-2p+1000, like you would normally do in your notebook; You must use the * (multiply) sign, otherwise you'll get a syntax error. Now we'll try to generate a simple sine waveform. Try this:

Amplitude(p)=sin(10)

Press [Preview] and… sorry, you still get nothing on the screen. The Wave Composer did not make a mistake! The sine of 10 in radians is exactly what it shows. You are unable to see the result because the line on your screen running across the 0 vertical point.

REMEMBER

The equation must be a function of a single variable and that variable must be directly related to the Horizontal axis Scale setting.

Now try this:

Amplitude(p)=sin(omg*p)

Still no good, but now press the [Adjust] button and here is your sinewave. So what's wrong? Well, if you'll give it a little amplitude it might help so, do it now exactly as follows:

Amplitude(p)=8000*sin(omg*p)

There you go. You should now see a perfect sine waveform with a period of 1000 points. This is because you have asked the Equation Editor to compute the sine along p points ("p" is the equation variable, remember?). If you want to create 10 sine waveforms, you should multiply p by 10. Try this:

Amplitude(p)=8000*sin(omg*p*10)

Equation Examples

So far, you have learned how to create two simple waveforms: straight lines and trigonometric functions. Let's see if we can combine these waveforms to something more interesting. Take the straight line equation and add it to the sinewave equation:

Amplitude(p)=12000*sin(omg*p*l0)-8*p+4000

Press [Preview]. Your screen should look like Figure 4-36.

Now let's try to modulate two sine waves with different periods and different start phase. Type this:

Amplitude(p)= 12000*sin(omg*p)*cos(omg*p*30)

Press [Preview]. Your screen should look like Figure 4-37.

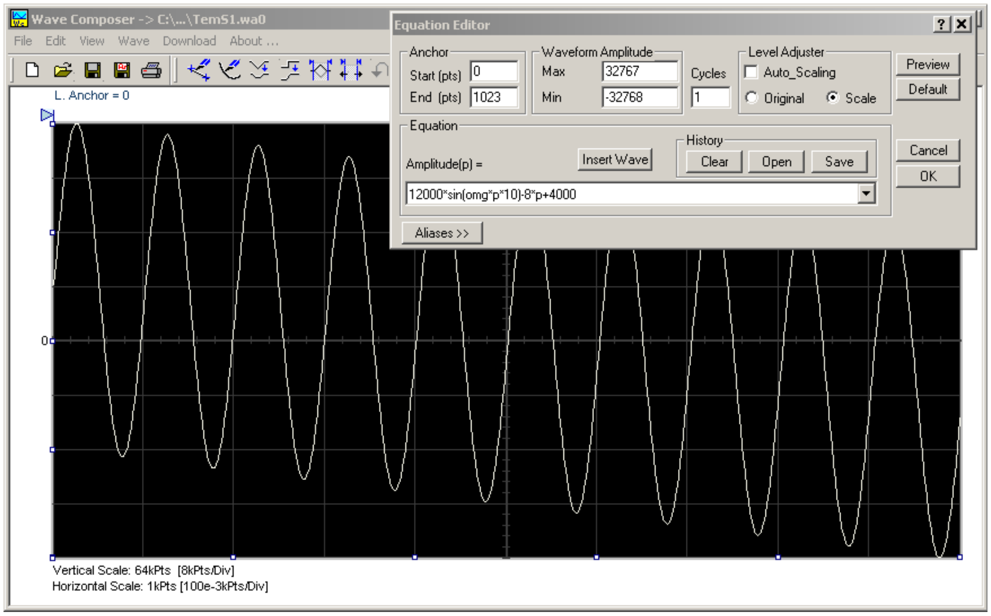

Figure 4-36, Equation Editor Example

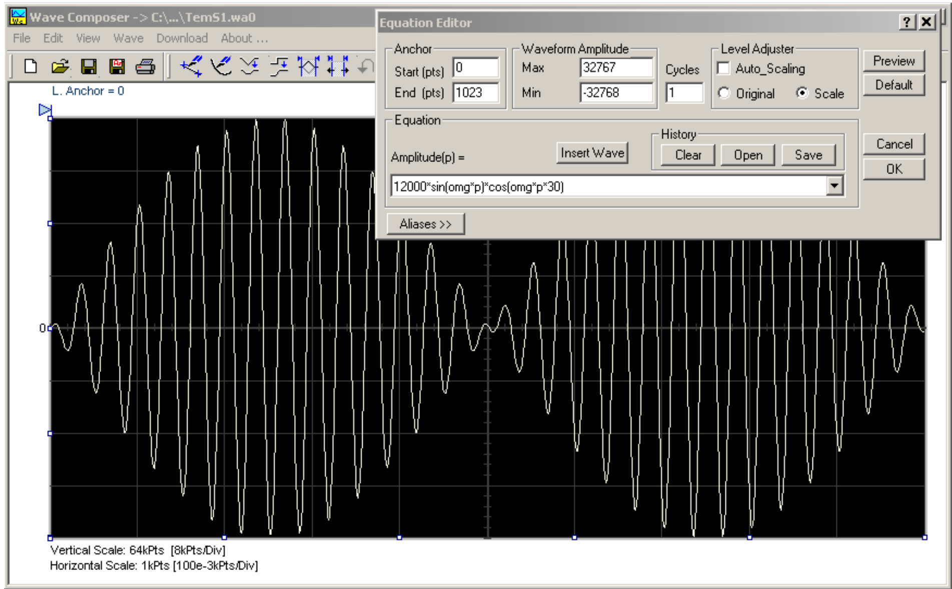

Figure 4-37, Using the Equation Editor to Modulate Sine Waveforms.

In the following example, 20% second harmonic distortion has been added to a standard sinewave. The original waveform had a peak-to-peak value of 24000 points so a 19% second harmonic is equivalent to 4500 points. The frequency of the second harmonic is obviously double that of the fundamental, so term +4500*sin(2*omg*p) is added to the original sine wave equation. Use the following equation:

Amplitude(p)=24000*sin(omg*p)+4500*sin(2*omg*p)

Press [Preview]. Your screen should look like Figure 4-38.

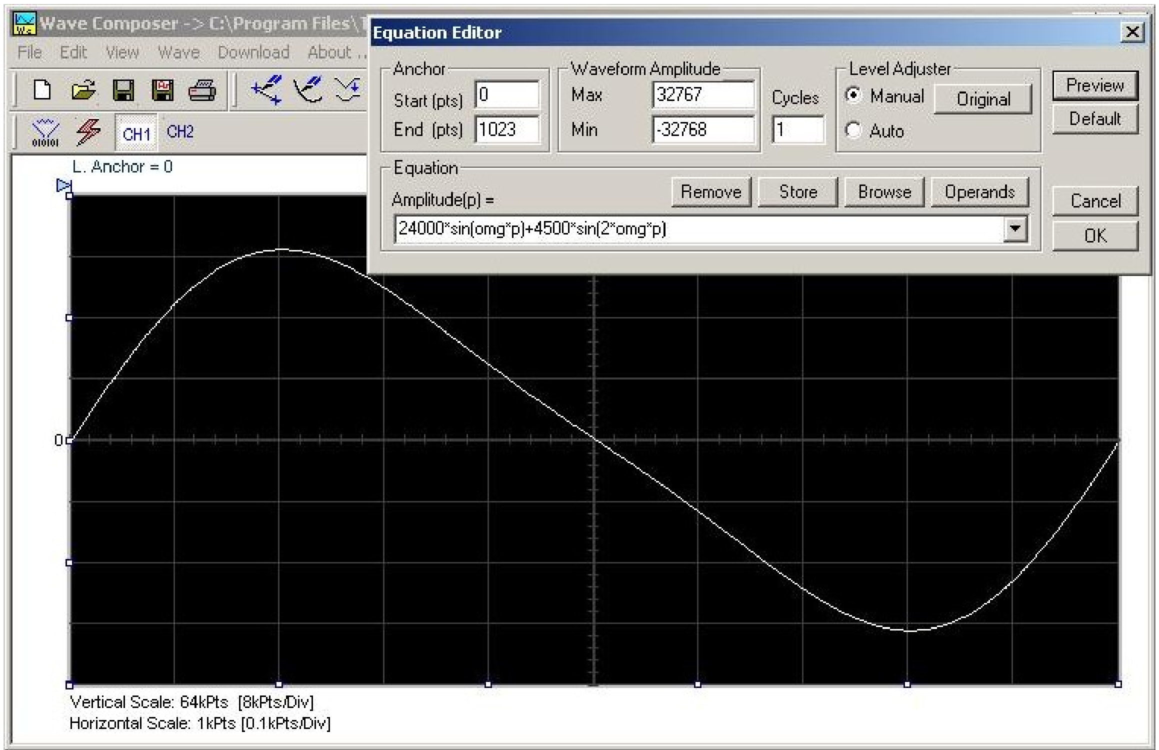

Figure 4-38, Using Equation Editor to Add Second Harmonic Distortion.

In Figure 4-40 we created 10 cycles of sinewave made to decay exponentially. The original expression for a standard sinewave is multiplied by the term e^(p/-250). Increasing the value of the divisor (200 in this case) will slow down the rate of decay.

Use the following equation:

Amplitude(p)=12000*sin(omg*p*10)*e^(p/-250)

Press [Preview] and [Accept] and the waveform graph should look like Figure 4-39.

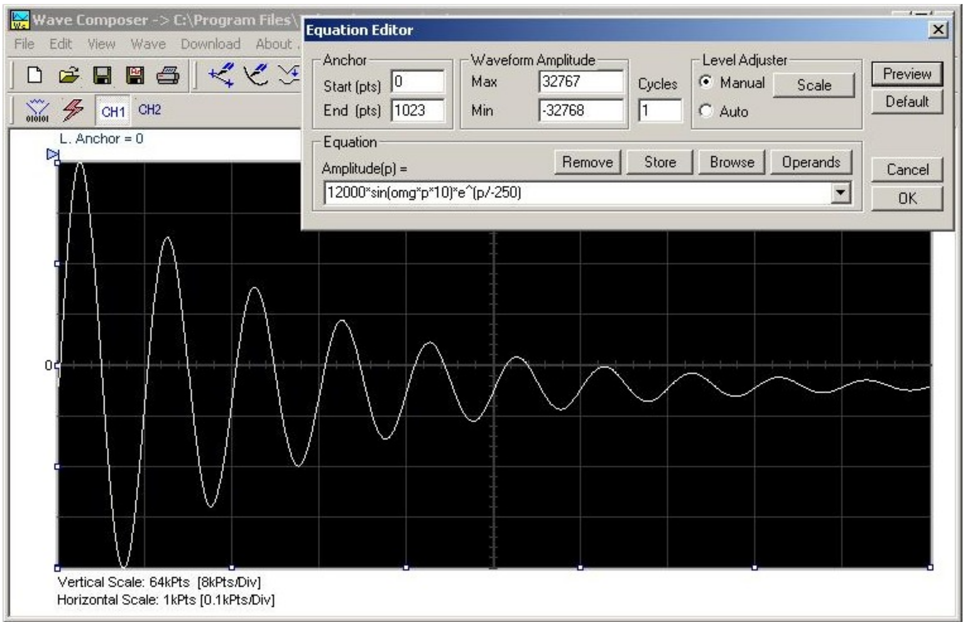

Figure 4-39, Using the Equation Editor to Generate Exponentially Decaying Sinewave

The last example as shown in Figure 4-39 is the most complex to be discussed here. Here, 100 cycles of a sine wave are amplitude modulated with 10 cycles of sine wave with a modulation depth of 20%. To achieve this, the upper and lower sidebands are defined separately and added to the fundamental or carrier. The upper sideband is produced by the expression 100*cos(110*omg*p) and the lower sideband by the term 100*cos(90*omg*p).

Use the following equation:

Ampl(p)=6000*sin(100*omg*p)+1200*cos(110*omg*p)-1200*cos(90*omg*p)

Press [Preview] and [Accept] and the waveform graph should look like Figure 4-40.

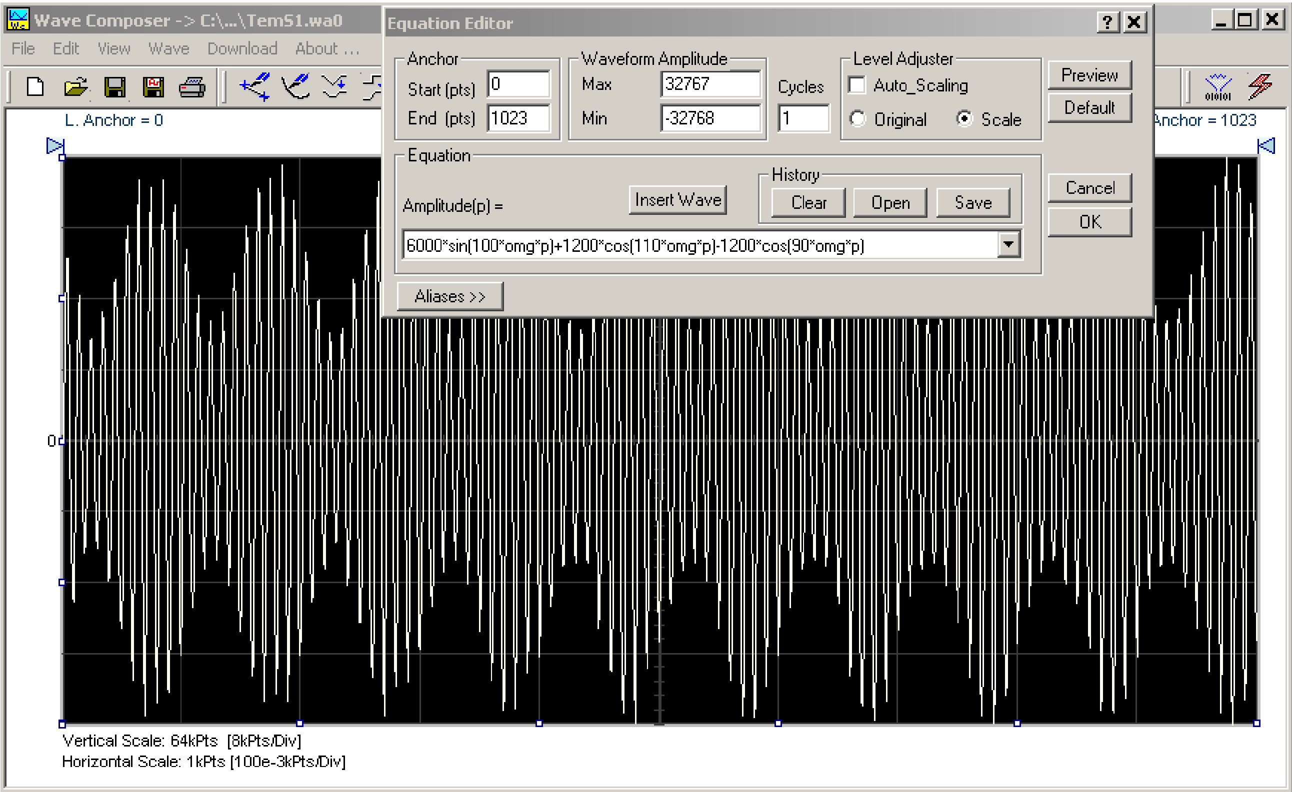

Figure 4-40, Using Equation Editor to Build Amplitude Modulated Signal with Sidebands

Combining Waveforms

The last feature to be described here allows you to combine waveforms which were previously stored in a file. You can write mathematical expressions that contain waveforms, simple operands, and trigonometric functions similar to the example given below. If you want to use stored waveforms in your equations, you must first generate these and store them in files. If you have stored files named Sine.wav and Noise.wav in the Wav12bit folder, you can enter them into your equation as shown.

Amplitude(p)= Sine.wav*sin(omg*p*10)+Noise.wav/1000

The above equation generates an amplitude modulated waveform with added noise. Note: You can also browse for .wav files stored in any folder using the Insert Wave button in the Equation Editor dialog box.

The following steps demonstrate how to create, store and combine waveforms using this example.

Step 1 – Create and store Sine.wav. Invoke the Wave command and generate a sine waveform. Press OK and then select the Save Waveform As… from the File command. Save this file into the default folder using the name Sine.wav in the default folder.

Step 2 – Create and store Noise.wav. From the Wave command select Noise. Click OK and watch your waveform screen draw a noise signal. From the File menu select Save Waveform As… and save this waveform into the default folder using the name Noise.wav.

Step 3 – Write and compute the original equation:

Amplitude(p)= Sine.wav*sin(omg*p*5)+Noise.wav/10

Press [Preview] and [Accept] and the waveform graph should look like Figure 4-41.

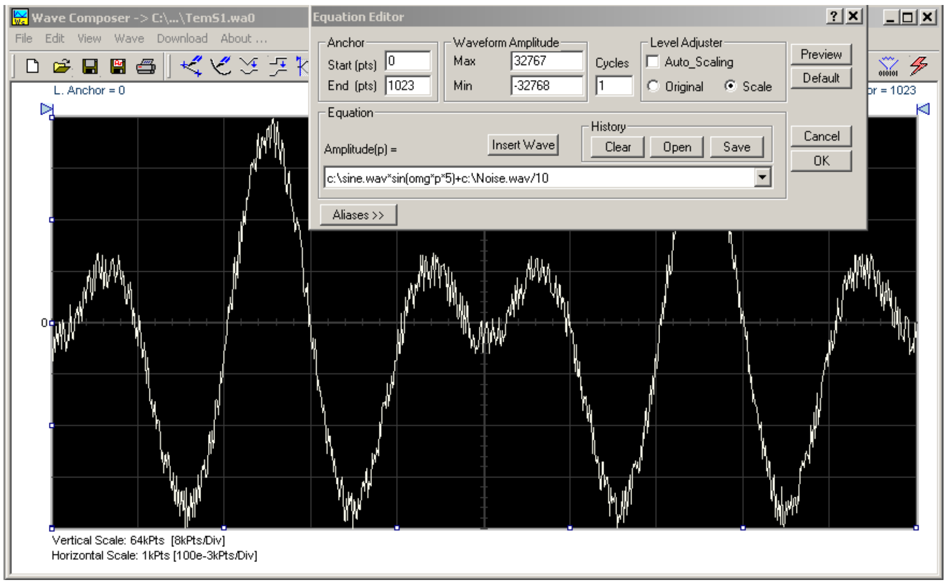

Figure 4-41, Combining Waveforms into Equations

The Pulse Composer

The Pulse Composer is a tool for creating and editing pulses without the need to think about sample clock, number of points and complex equations. Pulses are created on the screen, simply and efficiently in a special dialog box by typing in the width and level, or by using the "rubber band" method to place straight line segments with the exact amplitude and time duration. The Pulse Composer can also multiply pulse sections to create pulse duplication along lengthy time intervals.

When you finally have your pulse design on the screen the program determines if the pulse design will fit in one memory segment or use multiple segments and employ the sequence generator for repeatable segments. In either case, bear in mind that if you already have some waveforms stored in memory segments, these will be erased to make room for the new pulse design. If you insist on keeping arbitrary waveforms and still download complex pulses, you can check the "Force pulse to one segment" option and the 3172 will do some extra "muscle flexing" to fit the pulse as required.

To launch the Pulse Composer point and click on the Pulse tab in the Panels bar. Figure 4-42 shows an example of the Pulse Composer. The Pulse Composer has three main sections: Commands bar, Toolbar and Waveform screen.

Refer to Figure 4-42 throughout the descriptions to follow.

The Pulse Composer Menu Bar

The Pulse Composer menu bar provides access to standard Windows commands such as File and View. In addition, there are ArbConnection-specific commands such as Edit, Wave and System.

In general, clicking on one of the menus pulls down a list of commands. Clicking on a listed command may then either open a dialog box or generate an immediate action. For example, clicking on File and then Exit will cause the immediate termination of the Pulse Composer. The various commands in the Commands bar are listed and described below.

File Menu

The File menu has 4 menu items that control pulse waveform file operations. This menu also can be used to print the active waveform or to exit from Pulse Composer. Descriptions of the menu selections from the File pull-down menu are given below.

New

The New (Ctrl+N) menu item clears the pulse graph. Changes made to the pulse graph should be saved before using the New function because it is destructive to the displayed pulse.

Open…

The Open… (Ctrl+O) menu item lets you choose a previously saved pulse file and load it to the Pulse Composer graph. The *.PLS file extension, which is a text format, is supported by this operation.

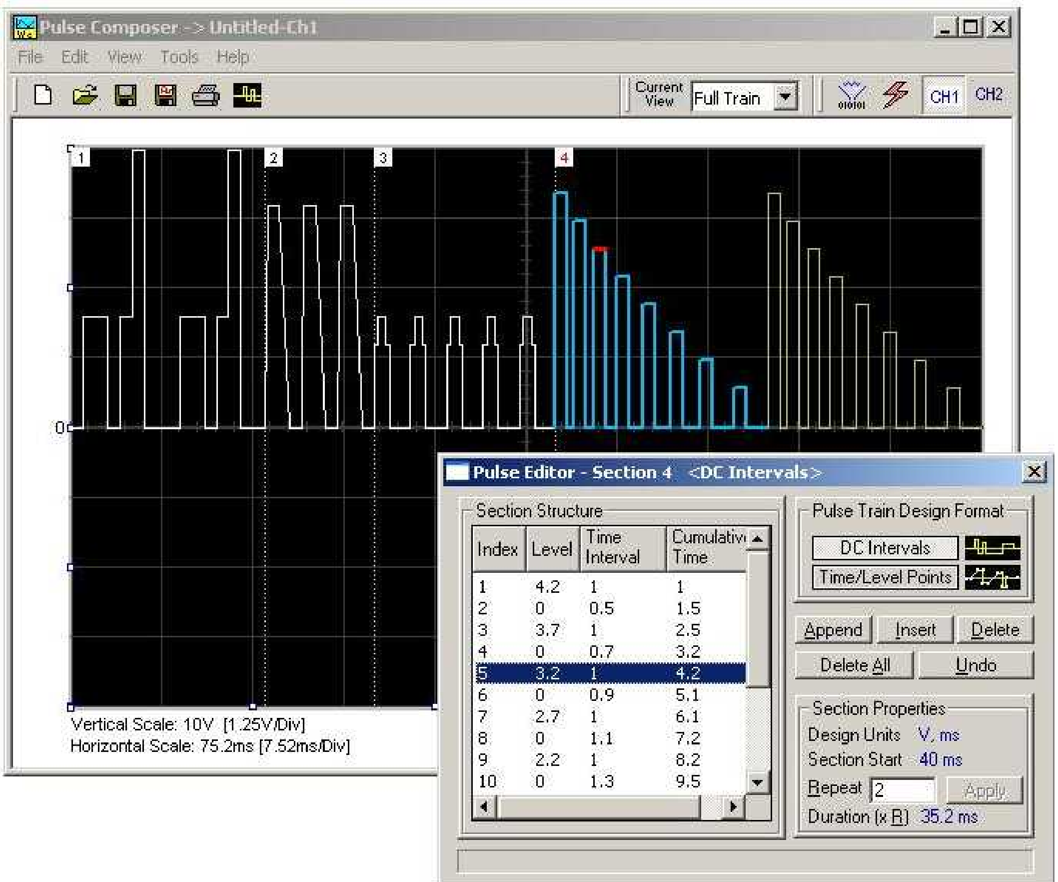

Figure 4-42, Pulse Composer Screen

Save

The Save (Ctrl+S) menu item lets you store your active waveform as a text file with a *.pls or *.wav extension. If this is the first time you save your pulse, the Save As… command will be invoked automatically, letting you select name, path, and format (*.pls or *.wav) for your pulse file.

Save As…
Use the Save As… menu item the first time you save your pulse. It will let you select name, location and format for your pulse file.

Print
With this menu selection you may print the active Pulse Window. The standard printer dialog box will appear and will let you select printer setup or print the waveform page.

Exit
The Exit menu item ends the current Pulse Composer session and takes you back to the Panels screen. If you made changes to your pulse since it was last saved, the Pulse Composer will prompt you to Save or Abandon changes these changes.

Edit Menu

The Edit menu is used for adding or removing pulse train sections. Use these commands to Append, Delete, Insert, or Undo last operation. The editing commands are explained in the following paragraphs.

Append Section
The Append Section menu command lets you append a new section at the end of the pulse train. Only one new section can be appended at the end of the train. If an empty section already exists, the append command will give an error. New sections are always appended at the end of the pulse train.

Insert Section
The insert Section menu command lets you insert a new section in between sections that were already designed. Only one new section can be inserted at the middle of the train. If an empty section already exists, the insert command will give an error.

Delete Section
The Delete Section menu command lets you remove sections from the pulse train without affecting the rest of the train. If you use this command from the Edit menu, make sure that the section you want to remove is currently the active section.

Remove all Sections
The Remove all Sections menu command lets you remove the entire pulse design from the pulse screen and start with a fresh page.

Undo
The Undo command reverses the last editing operation. This command is extremely useful in cases where you unintentionally delete a section from the pulse train and want to restore it to the pulse graph.

View Menu

The View menu lets you view various sections of the pulse graph. The View menu include: Pulse Editor, Full Train, Single Sections, and Options. Descriptions of the view menu items are given below.

Pulse Editor

The view Pulse Editor menu item invokes a dialog box as shown in Figure 4-43. In general, the Pulse Editor is used for placing straight line segments on the screen in intervals that define pulse width, rise/fall time, and amplitude. Information how to use the Pulse Editor to create pulse trains is given later in this chapter.

Full Train

The view Full Train menu item shows on the pulse graph all sections of the pulse train. Eventually, when all pulse sections have been designed, the entire pulse train as shown when the Full Train option has been selected will be downloaded to the instrument as a single waveform.

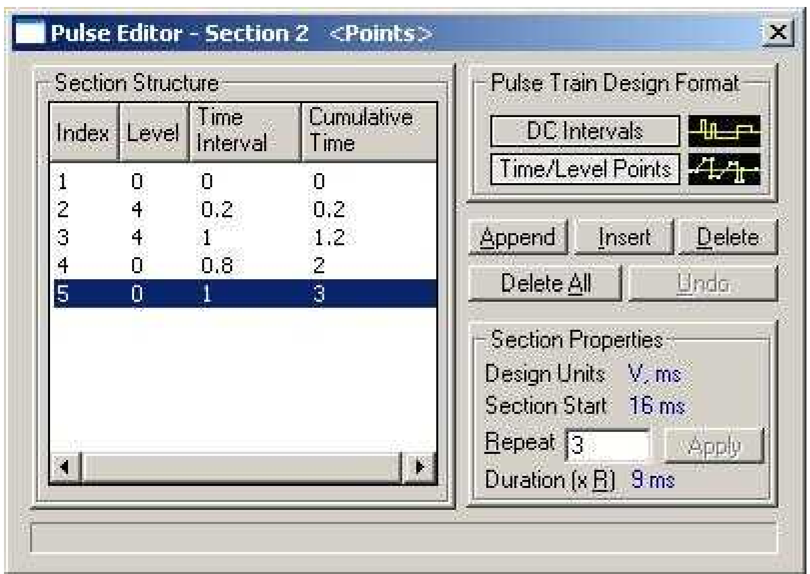

Figure 4-43, Pulse Editor

Single Section

The view Single Section menu item shows on the pulse graph one section at a time. Eventually, when all pulse sections have been designed, the entire pulse train as shown when the Full Train option has been selected will be downloaded to the instrument as a single waveform.

Options

The view options menu item opens the dialog box as shown in Figure 4-44. Use this dialog box to fine-tune the Pulse Composer to the way it should deal with operational modes and the waveform memory. Information on options is given later in this chapter.

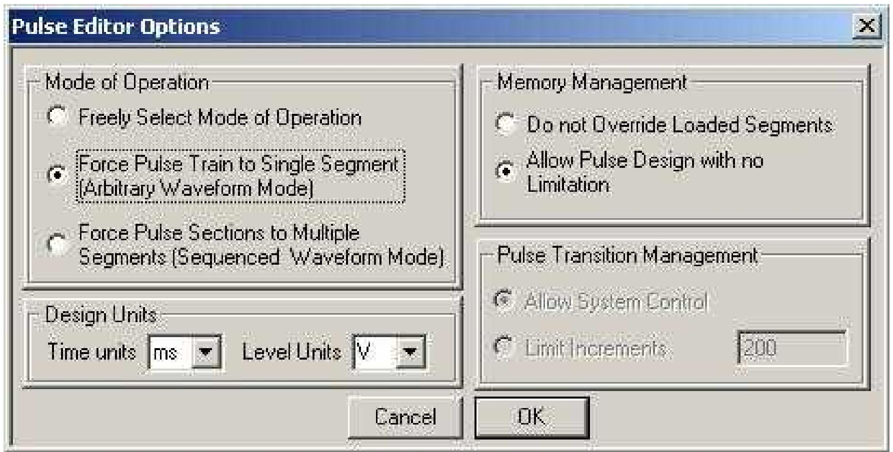

Figure 4-44, Pulse Editor Options

Tools Menu

The Tools menu lets you download pulse trains. You can also clear the entire pulse waveform memory using the Clear memory command.

Note

The Clear Memory command affects the entire waveform memory of the 3172. Be careful not to erase memory segments that you need to use and that haven't already been backed up.

The Pulse Composer Toolbar

The Pulse Composer toolbar (Figure 4-45) contains icons for editing the waveform graph, icons for saving and loading waveforms, fields for selecting an active channel, and more. The Toolbar is shown in the figure below. The icons, from left to right, operate the following functions: New waveform, Open an existing waveform file, Save pulse train, Save pulse train As, Print the screen, and open the Pulse Editor dialog box. Other icons select the current view on the screen, clear the memory and download the displayed pulse train to the active channel.

Figure 4-45, Pulse Composer Toolbar

Creating Pulses

As was mentioned above, creating pulses with the Pulse Composer is simple, intuitive, and can save you time when non-trivial pulses are needed. The Pulse Composer takes your design and processes the information, determines the appropriate instrument settings, and converts the pulse into a waveform for download to the instrument.

There are a number of terms that will be used throughout the following description. Make yourself familiar with these terms before attempting to use the Pulse Composer.

Pulse Editor

The Pulse Editor is the prime tool for creating pulses. To invoke the Pulse Editor, point and click on the Pulse Editor icon on the Pulse Composer toolbar. You can also invoke the editor by clicking on the Section Number icon as will be shown later in this description. The Pulse Editor dialog box is shown in Figure 4-43.

Pulse Train

The Pulse Train view exposes the entire pulse design. When downloading the waveform to the instrument, the entire pulse train is downloaded, regardless of the display mode

Pulse Section

A pulse train is constructed of 1 or more sections. If the pulse is simple, it can be created using one section only. For more complex pulse trains, the full train can be divided to smaller sections with each section designed separately. Figure 4-46 shows a complex pulse train which was made from five simpler sections and Figure 4-47 shows the design of the fifth section only of the pulse train.

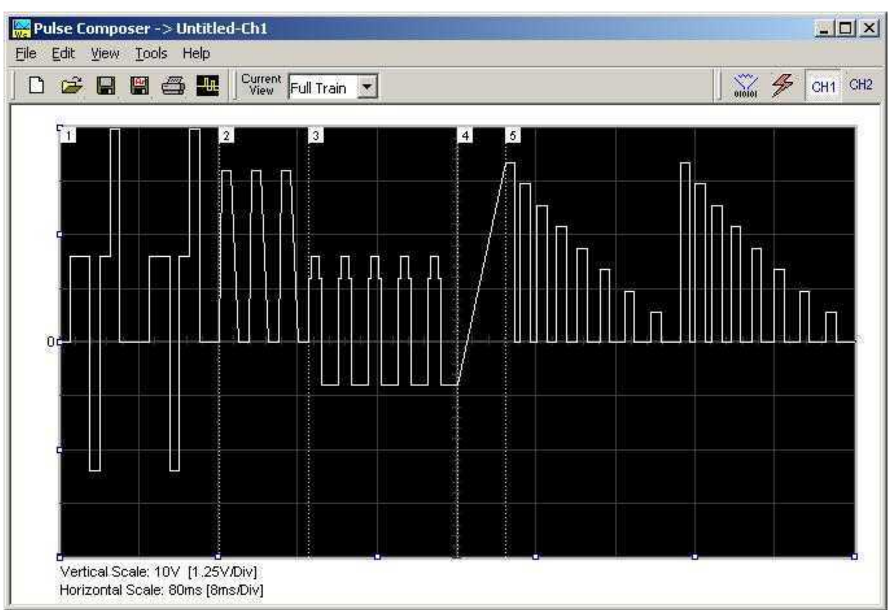

Figure 4-46, Complete Pulse Train Design

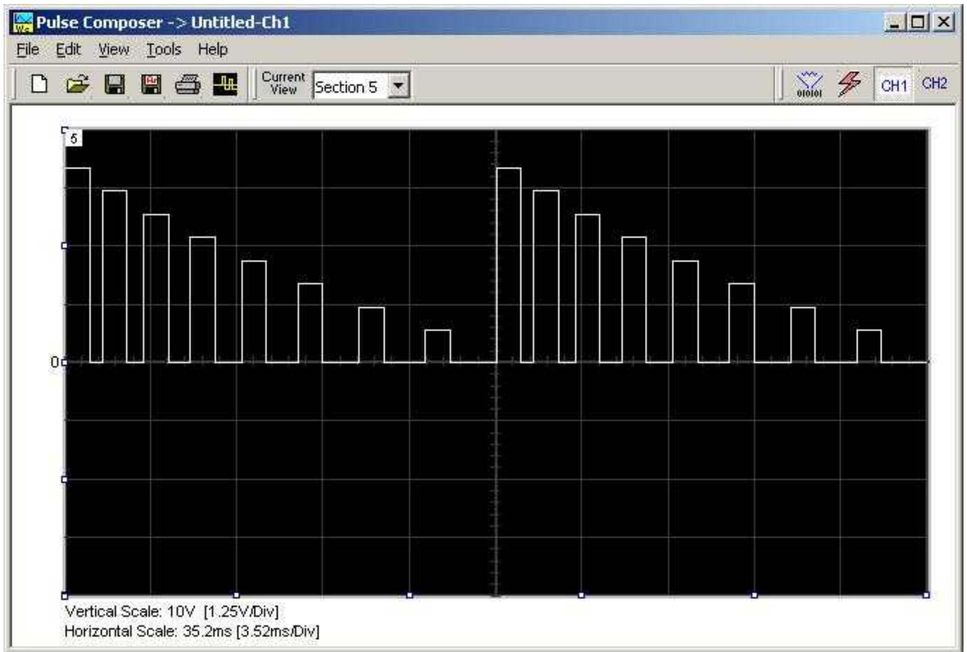

Figure 4-47, Section 5 of the Pulse Train Design

Now that we somewhat understand the Pulse Composer terminology, we can start with an example of how to design a pulse train like the one shown in Figure 4-47. If you already have some pulses shown on your Pulse Composer graph, click on New to start with a fresh page. Another initial step is to set the design parameters in the options menu to determine the way that the pulse will be stored in the 3172 waveform memory. Click on View→Options and refer to Figure 4-48 throughout the following description.

Setting the Pulse Editor Options

As shown in Figure 4-48, the Pulse Editor option dialog box is divided into four functional groups: Mode of operation, Design Units, Memory Management, and Pulse Transition Management. These groups are described below.

Mode of Operation
There are three options in the mode of operation group:

Freely Select Mode of Operation - use this mode of operation to let the generator decide for itself how to create pulses in 3172 waveform memory.

Force Pulse Train to Single Segment - recommended if you are using one pulse section only. In this case, the pulse waveform will occupy one segment only and the generator will automatically be set to operate in arbitrary mode.

Force Pulse Sections to Multiple Segments - places each section of the pulse train into a different memory segment and the generator will automatically be set to operate in sequenced mode. Select this option for the example we are going to build later.

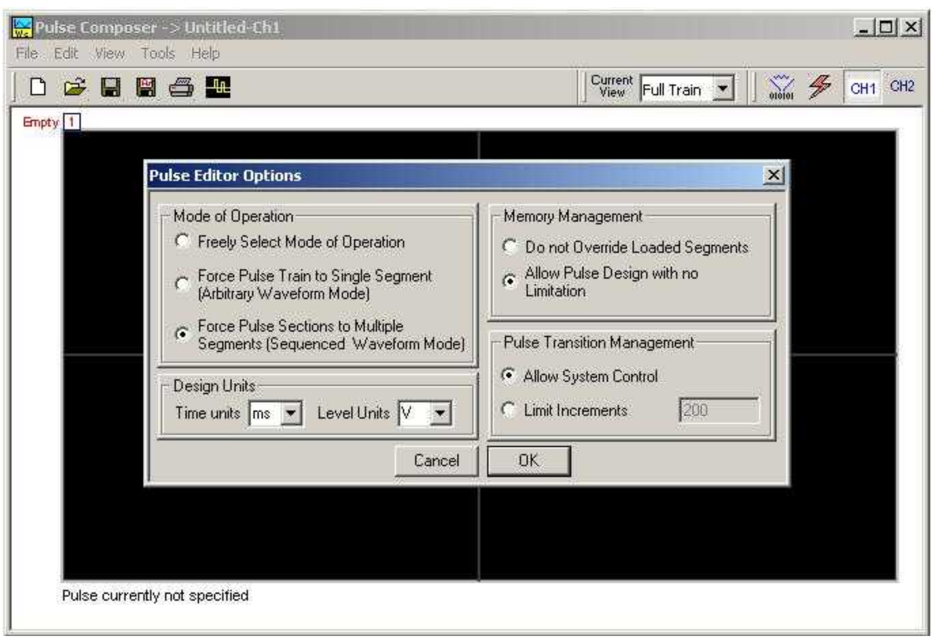

Figure 4-48, Selecting Pulse Editor Options

Design Units
As you design your pulse pattern, it is easiest to design it using the same engineering units as are used in your pulse specification.

Time Units - Select between μs, ms and s for the pulse interval

Level Units – Select between mV or V for the amplitude level.

The ms and V units will be used in the example to follow.

Memory management
There are two options in the memory management group.

Do Not Override Loaded Segments - makes sure that Pulse Composer does not overwrite waveforms already stored in memory.

Allow Pulse Design With No Limitations - allows Pulse Composer to overwrite waveforms already stored in memory.

Pulse Transition management
The pulse transition management parameter defines for the program how many waveform points will be used to step from one amplitude level to another amplitude level. The longer the transition time, the more steps the program will need to smooth the transition.

Allow System Control – Lets the 3172 decide how to make the transitions efficient in terms of memory usage and slope smoothness.

Limit Increments – Lets you manually control how many waveform points are used in transitions.

After you complete setting the Pulse Editor options, click on OK.

Using the Pulse Editor

The prime tool for building pulse patterns on the Pulse Composer screen is the Pulse Editor. To invoke the Pulse Editor, click on the Pulse Editor icon on the Pulse Composer toolbar and the editor shown in Figure 4-49 appears. Refer to this figure for the following descriptions.

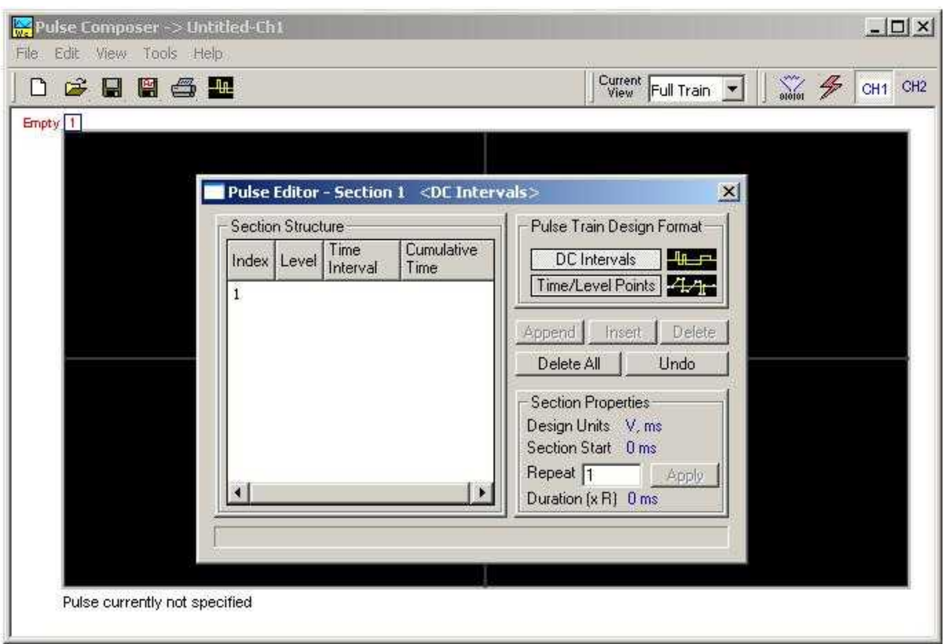

Figure 4-49, Using the Pulse Editor

The Pulse Editor has four groups: Section Structure, Pulse Train

Design Format, Section Properties, and control buttons. These groups are described below.

Pulse Train Design Format

There are two methods (or formats) that can be used for designing the pulse shape: DC Intervals and Time/Level Points. The design format is unique for the current section and cannot be switched in the middle of a pulse section design.

DC Intervals – programs pulse duration using DC levels only. Transition times for this format are at the maximum rate that the generator can produce. For example, if you want to draw a simple square waveform that has a 0V to 3.3V amplitude, a 50% duty cycle, and a 1ms period, you enter the following parameters:

Index = 1, Level = 3.3, Time interval = 0.5 (Cumulative Time = 0.5)
Index = 2, Level = 0, Time Interval = 0.5 (Cumulative Time = 1.0)

Note that as you build the segments that the pulse is being drawn on the screen as you type in the parameters. Also note that the Cumulative Time column is updated automatically with the cumulative time lapse from the start of the pulse.

Time/Level Points – programs pulse turning points using level and time markers. This format is a bit more complex to use, however, it allows the design of pulses with linear transition times. For example, if you want to draw a simple square waveform that has a 0V to 3.3V amplitude, a 50% duty cycle, a 1ms period and a 100ns transition time, you enter the following parameters:

Index = 1, Level = 0, Time interval = 0, (Cumulative Time = 0)
Index = 2, Level = 3.3, Time Interval = 0.1, (Cumulative Time = 0.1)
Index = 3, Level = 3.3, Time interval = 0.4, (Cumulative Time = 0.5)
Index = 4, Level = 0, Time interval = 0.1, (Cumulative Time = 0.6)
Index = 5, Level = 0, Time interval = 0.4, (Cumulative Time = 1.0)

Note that as you build the segments that the pulse is being drawn on the screen as you type in the parameters and the specified point is marked with a red dot. Also note that the Cumulative Time column is updated automatically with the cumulative time lapse from the start of the pulse.

Section Structure

The term Section Structure is used to define the pulse train's common properties. There are four parameters that can be programmed in this group: Index, Level, Time Interval and Cumulative Time.

Index – Is added automatically as you program pulse segments. The index line is highlighted as you point and click on pulse segments on the Pulse Editor screen.

Level – Specifies that peak level of the programmed segment. As you build the pulse, the level window is expended automatically to fit the required amplitude range.

Time Interval – Specifies the time that will lapse for the current index level. You can program the time interval and the cumulative time will be adjusted accordingly.

Cumulative Time – Specifies the time that will lapse from the start of the current pulse section. You can program the cumulative time and the time interval will be adjusted accordingly.

Section Properties

The Section Properties contains a summary of properties that are unique for the current section.

Design Units – Provides information on the units that are used when you draw the pulse segments. These units can be changed in the Pulse Editor options.

Section Start – Provides timing information for the start of the current section. If this is the first pulse section the value will always be 0. Subsequent sections will show the start mark equal to the end mark of the previous section.

Repeat – Allows multiplication of pulse segments without the need to re-design repetitive parts. After you enter a repeat value, press the Apply button to lock in the repeat multiplier.

Duration – Displays the time that will lapse from the start of the pulse section to the end. The duration shows the total time lapse, including the repeated sections.

Control Buttons

The control buttons allow you to append, insert, or delete one or all index lines. The Undo button is useful in cases where an error was made and restoration of the last operation is critical.

Pulse Example, Section 1

Now that we are familiar with the Pulse Composer and its operation, we are ready to start building the first section of the pulse as shown in Figure 4-50. Point and click on the New icon and open the Pulse Editor. Type in the level and time intervals as shown. Note that the pulse segments are being created on the screen as you type the values.

Tips

1. Use the tab key to navigate Section Structure fields.

2. Use Append to add an index line at the end of the list.

3. Use Insert to add a segment above the selected line.

Before we proceed with the design of the next section, observe some values which are displayed on the Pulse Composer screen. In the lower left corner of the composer, the Vertical Scale is 10 V (1.25 V/Div) and the Horizontal Scale is 14 ms (1.4 ms/Div). These two values are critical for the integrity of the design because they will later be used by the program to set pulse timing. These values can change as you add more sections to the pulse train.

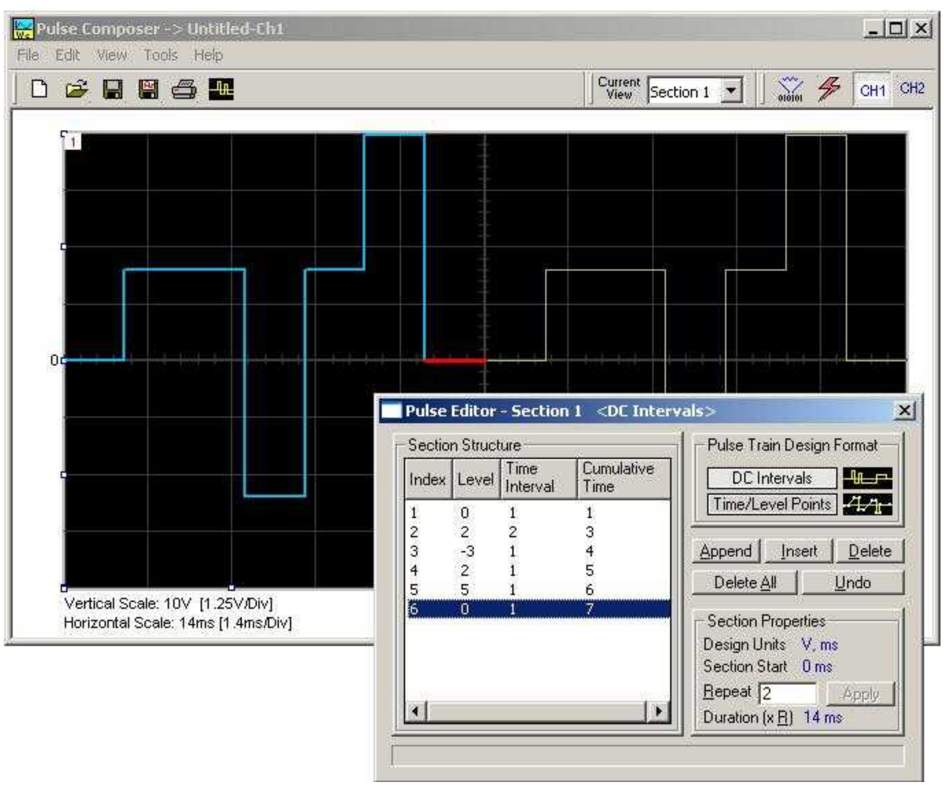

Figure 4-50, Building Section 1 of the Pulse Example

Pulse Example, Section 2

The first pulse section is complete. We are ready now to start building the second section of the pulse as shown in Figure 4-51. Use the Pulse Composer's Edit menu to select the Append Section operation. A new section number will appear but its fields will be initially empty to the right of the section identifier.

Before you start entering values in this section, note that there are linear transitions required for this section. Therefore, select the Time/Level Points option in the Pulse Train Design Format group. You are now ready to start programming values. If you try to switch design formats after you have already typed in some values, the Pulse Editor will display a warning box alerting you that the design format can only be changed for an empty section. In this case, the only way to recover is to delete all entries and start from an empty index list. Type the section entries as shown in the figure.

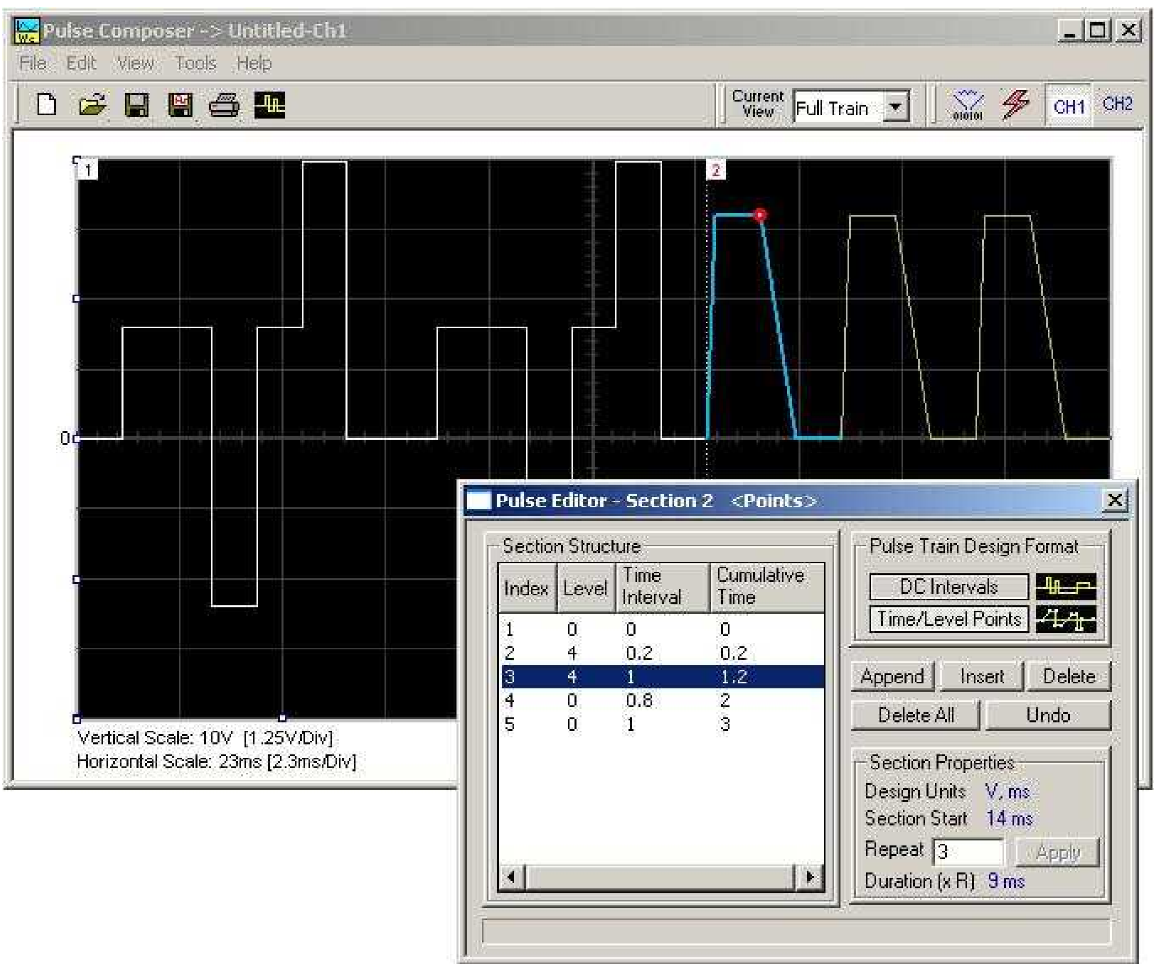

Figure 4-51, Building Section 2 of the Pulse Example

Pulse Example, Section 3

The second pulse section is now complete. We are ready now to start building the third section of the pulse as shown in Figure 4-52. Use the Edit menu to select the Append Section operation. A new section number will appear but its fields will be initially empty to the right of the section identifier.

Before you start entering values to this section, note that there are fast transitions required for this section. Therefore, select the DC Intervals option in the Pulse Train Design Format. You are now ready to start programming values. If you try to switch design formats after you have already typed in some values, the Pulse Editor will display a warning box alerting you that the design format can only be changed for an empty section. In this case, the only way to recover is to delete all entries and start from an empty index list. Type the section entries as shown in the figure.

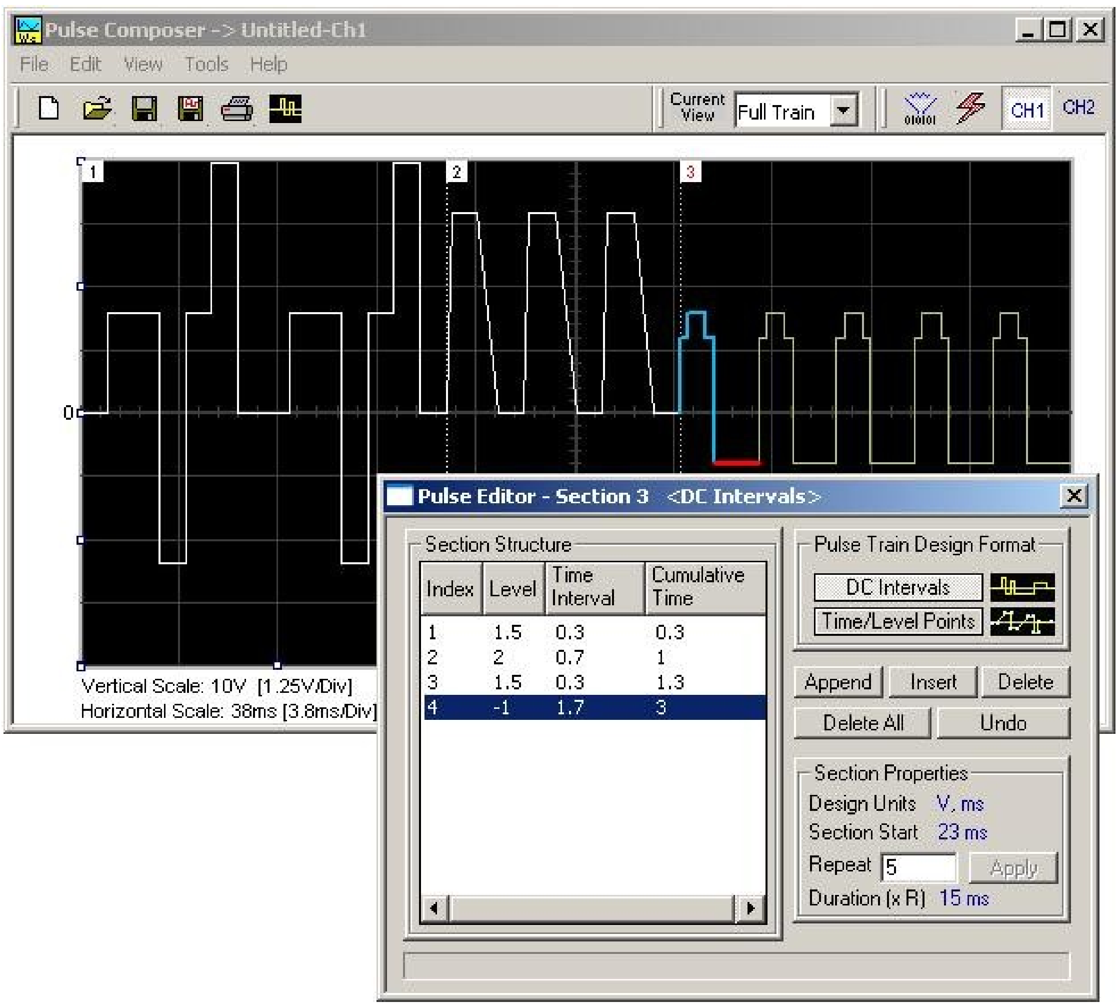

Figure 4-52, Building Section 3 of the Pulse Example

Pulse Example, Section 4

The third pulse section is now complete. We are ready now to start building the forth section of the pulse as shown in Figure 4-53. Use the Edit menu to select the Append Section operation. A new section number will appear but its fields will be initially empty to the right of the section identifier.

Before you start entering values into this section, note that there is only one linear transition required for this section that will start from the last point of the previous section and will connect to the start point of the next section. Therefore, select the Time/Level Points option in the Pulse Train Design Format. You are now ready to start programming values. Type the section entries as shown in the figure.

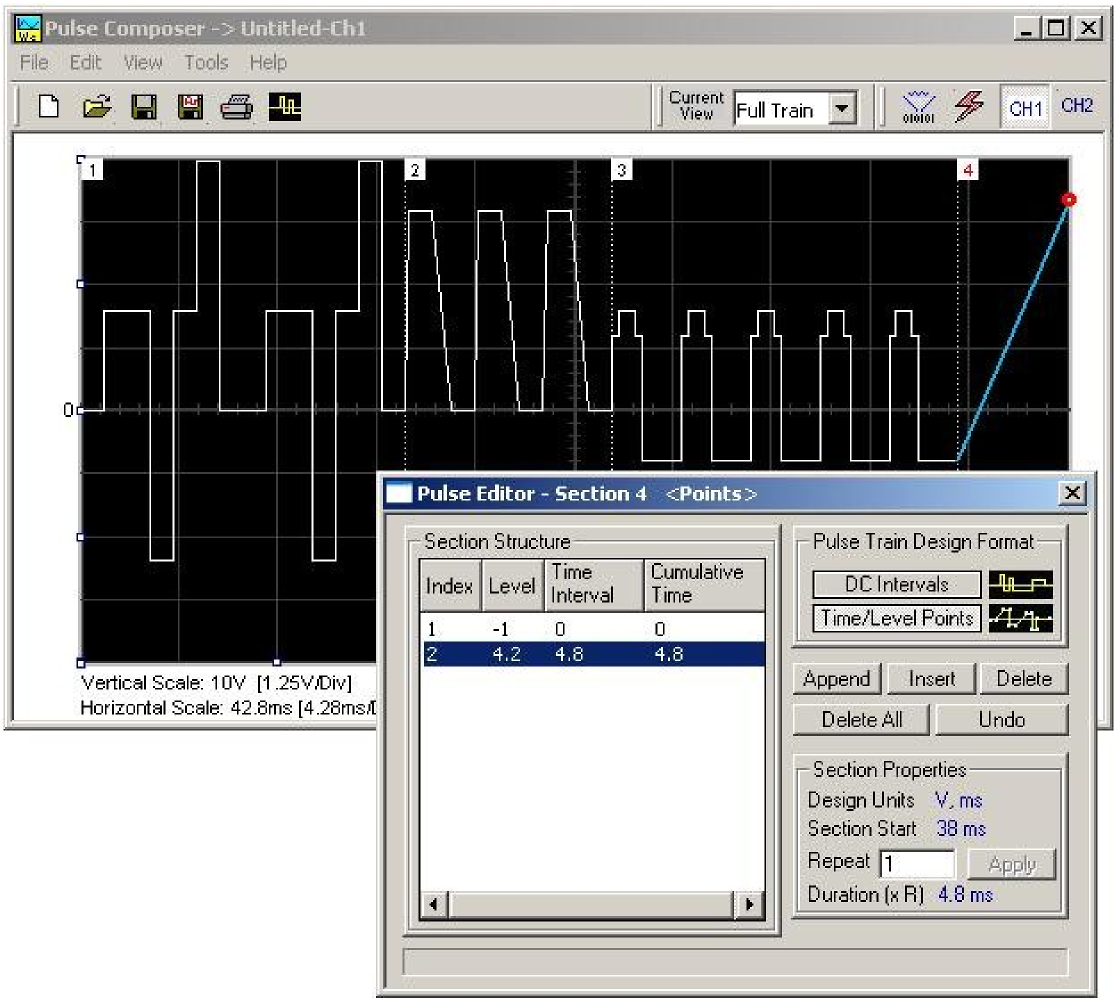

Figure 4-53, Building Section 4 of the Pulse Example

Pulse Example, Section 5

The fourth pulse section is complete so we are now ready to start building the fifth and final section of the pulse as shown in Figure 4-54. Use the Edit menu to select the Append Section operation. A new section number will appear but its fields will be initially empty to the right of the section identifier.

Note that there are fast transitions required for this section that will start from the last point of the previous section and will connect to the start point of the next section. Therefore, select the Time/Level Points option in the Pulse Train Design Format. You are now ready to start programming values. Type the section entries as shown in the figure.

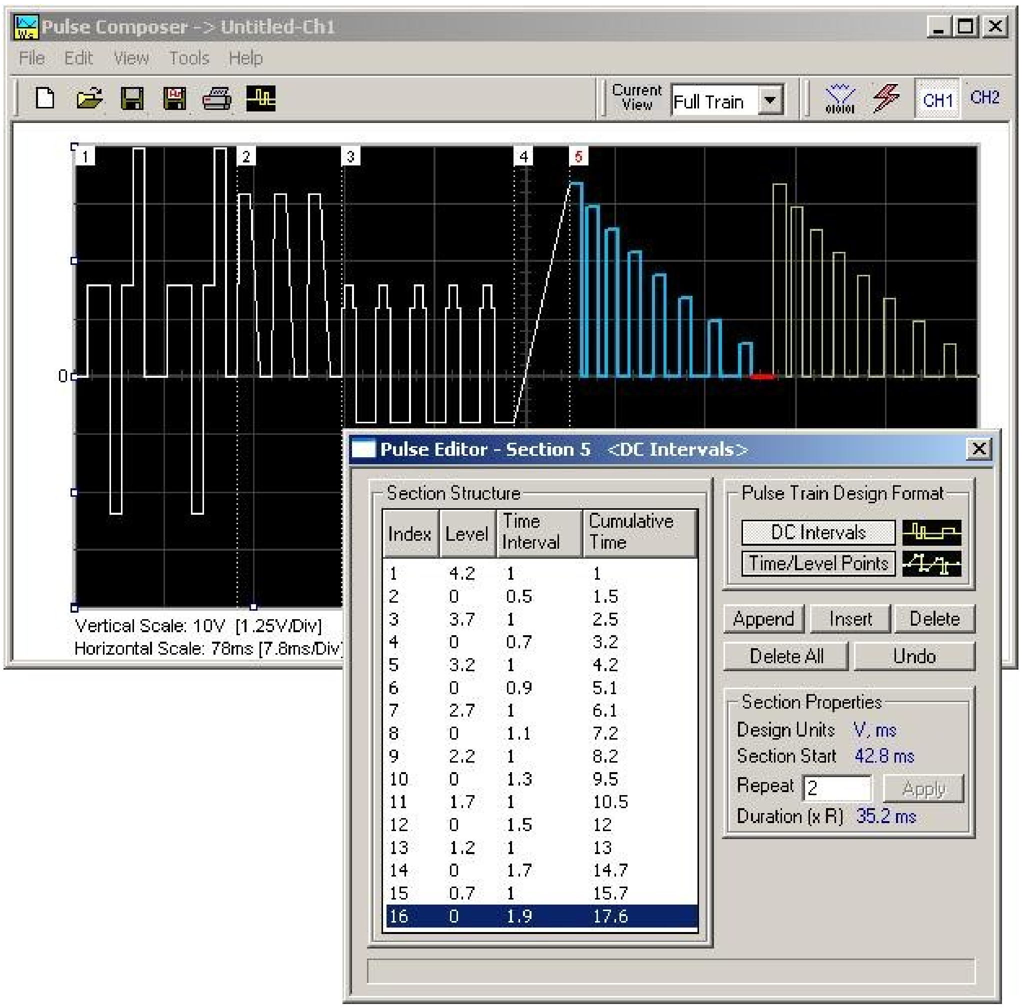

Figure 4-54, Building Section 5 of the Pulse Example

Downloading the Pulse Train

If you followed the above description to build this pulse example, the screen should look as shown in Figure 4-55. The next step is to download what you see on the Pulse Composer graph into the 3172 waveform generator.

One last step before you download the waveform to the instrument is to check the Pulse Train Download Summary which appears after you click on the Download icon on the Pulse Composer toolbar. Refer to Figure 4-55 for the next section on how to interpret the download summary.

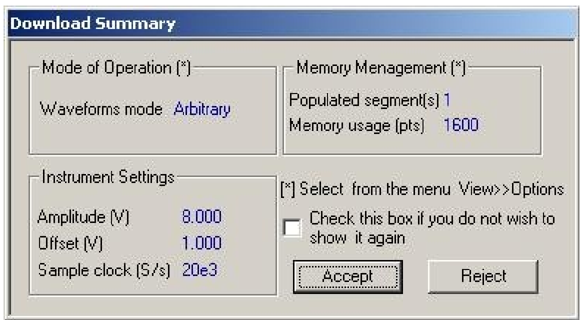

Figure 4-55, Pulse Editor Download Summary

Interpreting the Download Summary

It is important to understand that when you download a pulse waveform from the Pulse Composer, the parameters and mode of operation of the 3172 might be altered. The download summary shows what the new mode of operation will be so that you can reject the new settings if you do not agree to the changes. Once you press the Accept button, the waveform will be downloaded to the generator and the modes and parameters will be updated as shown in the dialog box.

If you are already familiar with the changes and do not wish to see the download summary every time you download a pulse waveform, you can check the box and it will not be shown on future downloads. You can restore this summary by selecting the View>>Download Summary menu item.

Mode of Operation – This describes mode of operation setting to be used after completion of the pulse download. This field could display one of two options: Arbitrary or Sequenced. Pay attention to the note (*) that says "Select from the menu View>>Options" Since, for this example, we checked the Force Pulse Train to Single Segment (see Figure 4-50), so that the generator forces the waveform mode to be Arbitrary and thus only one segment can be loaded with the pulse train.

Memory Management – By selecting the arbitrary mode of operation, the pulse train is forced to a single segment. This summary shows which segment has been populated and how much memory is needed to build the required pulse train.

Instrument Settings – Shows the amplitude, offset, and sample clock settings that will be changed on the generator. The settings in this summary cannot be affected from the Pulse Editor options settings. These are computed and modified automatically for the current pulse train pattern and will change from pattern to pattern.

Accept/Reject – These buttons are the final checks before you download the pulse train to the instrument. If you are unhappy with the instrument setting and want to change some of the options, there is still time click on the Reject button and do more changes. Click on the Accept button to complete the download process.

The FM Composer

The FM Composer looks and feels almost like the waveform composer except there is a major difference in what it does. If you look at the opening screen as shown in Figure 4-56, you'll see that the vertical axis is marked with frequencies. You'll see later that as you draw waveforms on the FM composer screen, these waveforms represent frequency changes and not amplitude changes as are generated by the waveform composer.

The FM composer is a great tool for controlling frequency agility by generating the agility curve as an arbitrary waveform. For example, if you create a sine waveform, the 3172 will generate frequency-modulated signal that will follow the sine pattern. The resolution and accuracy of the modulated waveform is unsurpassed and can only be duplicated by mathematical simulation. The FM composer is loaded with many features and options so use the following paragraphs to learn how to create and download modulating waveforms to the 3172 using the FM Composer.

Invoke the FM Composer from Panels bar. The Wave Composer has three sections: Commands bar, Toolbar and Waveform screen. Refer to Figure 4-56 throughout the description of these parts.

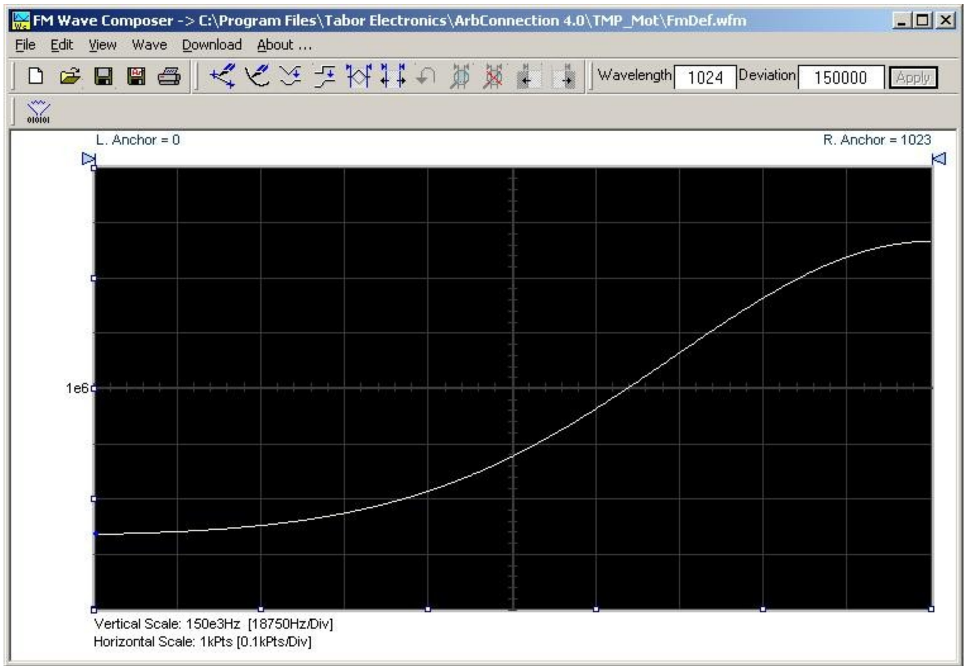

Figure 4-56, FM Composer Opening Screen

The Menu Bar

The FM Composer menu bar is an exact duplication of the menu bar of the Wave composer. It provides access to standard Windows commands such as File and View.

File Menu

The File menu has 4 menu selections which that control waveform file I/O operations. Also use this menu to print the waveform or to exit the FM Composer program. Description of the various commands under File is given below.

New Waveform
The New Waveform command will remove the waveform from the screen. If you made changes to the waveform area and use this command, you should save your work before clearing the screen. The New Waveform command is destructive to the displayed waveform.

Open Waveform…
The Open Waveform… menu item lets you browse for previously saved waveform files and to load these waveforms to the waveform graph. This command is also very useful for converting waveform files to FM Composer format files (*.wvf).

Save Waveform
The Save Waveform menu item stores your active waveform as a binary file with a *.wvf extension. If this is the first time that you save this FM waveform, the Save Waveform As… command will be invoked automatically, letting you select name, path, and format for the waveform file.

Save Waveform As…
Use the Save Waveform As… menu item the first time you save your waveform. It will let you select name, location and format for your waveform file.

Print
With this command you may print the active Waveform Window. The standard printer dialog box will appear and will let you select printer setup, or print the waveform page.

Exit
The Exit command ends the current FM Composer session and takes you back to the Panels screen. If you made changes to your waveform since it was last saved, make sure to Save your work before you use this command.

Wave Commands

The Wave menu lets you create waveforms on the waveform graph. The Wave menu has a library of 6 waveforms: Sine, Triangle, Square, Exponent, Pulse, and Noise. It also lets you create waveforms using an Equation editor. Information how to create waveforms using the Wave menu is given below.

Creating Waveforms From the Built-in Library

You can create any waveform from the built-in library using the Wave menu. Clicking on one of the Wave options will open a dialog box. An example of the Sine waveform dialog box is shown in Figure 4-55. This dialog box is representative of the rest of the waveforms, so other waveforms will not be described.

Creating Sine Waveforms

Use the following procedure to create sine waveforms from the built-in library. Click on Wave, then sine… The dialog box shown in Figure 4-57 appears. You can now start programming parameters that are available in this box.

Start Point Anchor – Defines the first point where the created wave starts. Note that if you change the start point the left anchor automatically adjusts itself to the selected start point. The example shows start point set at point 200.

End Point Anchor – Defines where the created waveform will end. Note that as you change the end point the right anchor will automatically adjust itself to the selected end point. The example shows end point set at point 499.

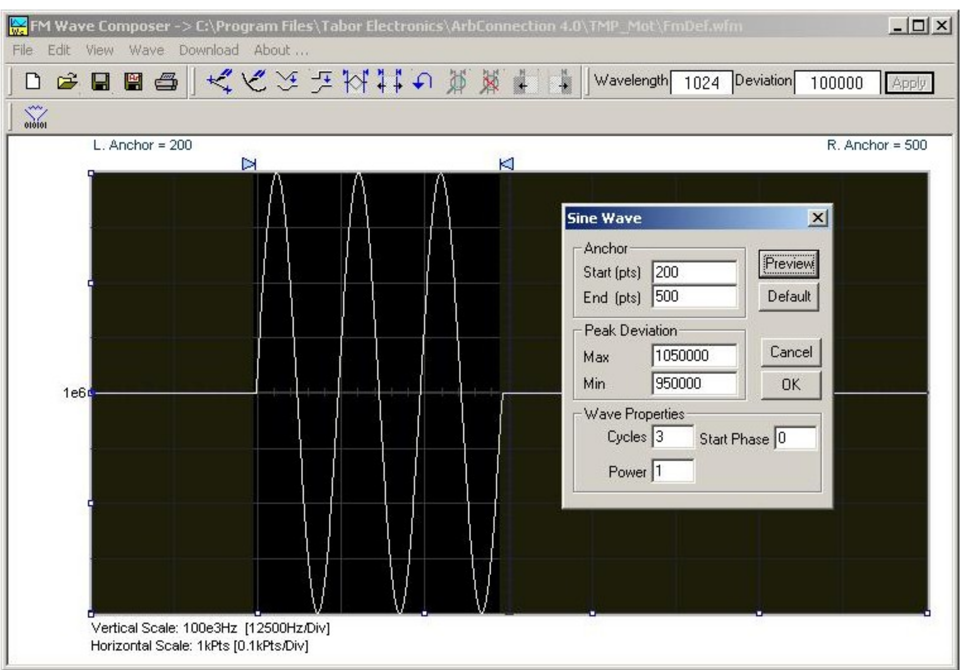

Figure 4-57, Generating Sine Modulation Using the FM Composer

Max. Peak Deviation – This parameter defines the forward peak deviation. Note that the forward peak deviation cannot exceed the pre-defined Deviation parameter as shown on the Toolbar. In case you need to exceed the pre-defined peak value you must quit this box and modify the Deviation parameter to provide sufficient range for the forward peak deviation range.

Min. Peak Deviation – This parameter defines the backwards peak deviation. Note that the backwards peak deviation cannot exceed the pre-defined Deviation parameter as shown on the Toolbar. In case you need to exceed the pre-defined peak value you must quit this box and modify the Deviation parameter to provide sufficient range for the backwards peak deviation range.

Cycles – The Cycles parameter defines how many sine cycles will be created within the specified start and end anchor points. The example below shows three sine cycles.

Start Phase – The start phase parameter defines the angle at which the sine will start. The example shows 0° start phase.

Power – Sine to the power of 1 will generate a perfect sine. Power range is from 1 through 9.

Tip

The functionality of the FM composer is similar to the Wave composer. If you need more information on the FM Composer functions and Equation Editor, refer to the Wave Composer section in this manual.

The 3D Composer

The 3D Composer was specifically designed for simultaneous profiling of amplitude, frequency and phase. Amplitude profiles can be designed separately for channels 1 and 2, but frequency and phase profiles are shared by both channels. The following paragraphs will describe the various sections of the 3D composer and will guide you through some 3D programming examples.

The opening screen of the 3D composer is shown in Figure 4-58. As you can see it does not at all look like any of the other composers that were described previously discussed however, generating waveforms and programming profiles is very similar to other composer so you will be up and running in no time.

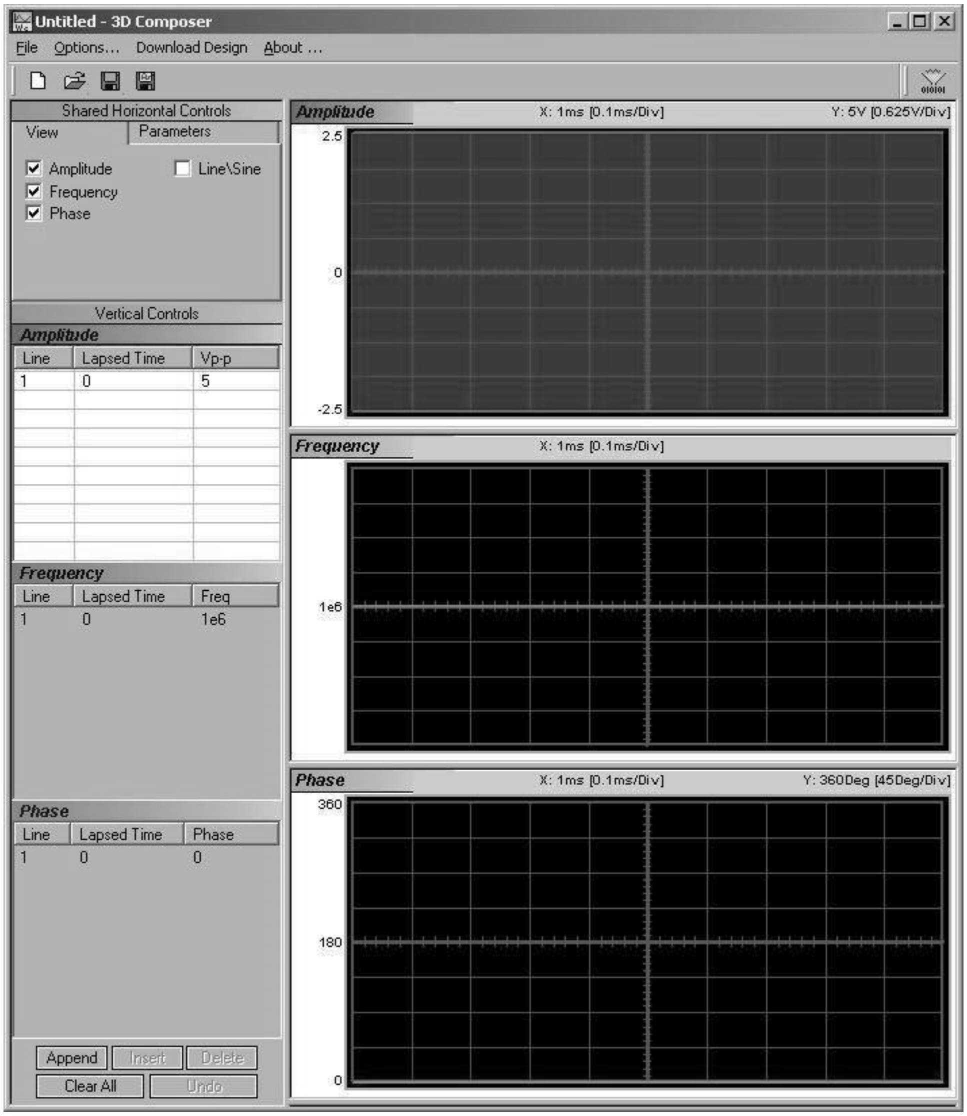

Figure 4-58, 3D Composer Screen

The 3D composer has three main sections: Shared horizontal Controls, Vertical Controls and Graphical Screens. The panels on the left are used for designing the waveform parameters and the screens on the right side depict the shape of the profile. Below find a detailed description of all of these sections. Refer to Figure 4-58 throughout the description.

Shared Horizontal Controls

The *Shared Horizontal Control* has two tabs: *View* and *Parameters*.

View

The *View* tab is useful if you are interested in programming 1 or two profiles only and do not care to see other screens. Check the boxes for the profiles you wish to program only and these will be shown on the screen. For example, if you check the Amplitude and the Frequency options, the Phase screens will not be visible.

Parameters

The *Parameters* tab, as shown in Figure 4-59, is used for setting up the duration of the signal, the position of the marker (if required) and the amount of memory that is allocated for this purpose. Setting up correctly the parameters in this group is the basic and the most important task before you start designing 3D waveforms. The duration can be set in units of ns, us, ms, and seconds and can be programmed within the range of 800 ns to 30,000 s.

The 3D profiler behaves just like an arbitrary waveform. The shape of the profiler is generated using waveform points and a dedicated 3D sample clock. So, just as the basics for an arbitrary waveform design, the duration is derived from the following relationship:

Duration = SCLK / # of waveform points

where SCLK is the 3D sample clock and the # of waveform points can be programmed from 2 to 30,000.

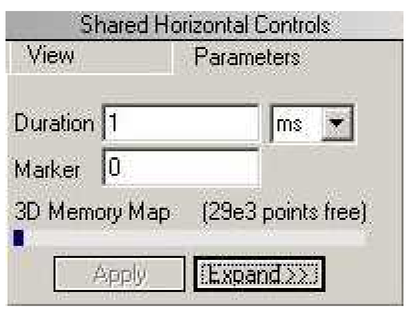

Figure 4-59, Parameters Tab

The recommended method is to let the 3D composer set up the sample clock and the numbers of points automatically for you, however, in some cases you may want to fine tune your requirement by pressing the Expand button. Figure 4-60 shows the Expanded Parameters options dialog box.

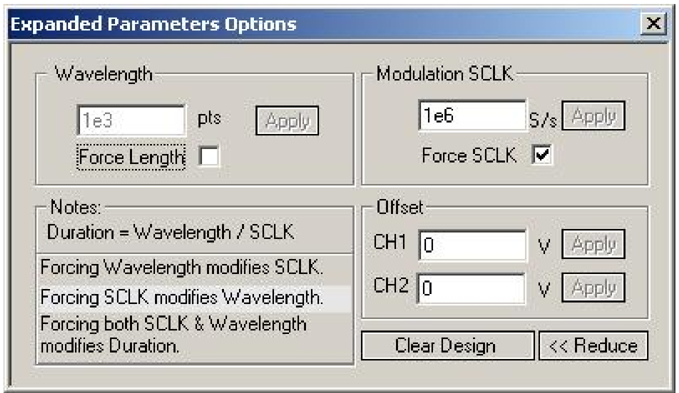

Figure 4-60, Expanded Parameters Options Dialog Box

The Expanded Parameters options dialog box has three sections: Wavelength, Modulation SCLK and Offset. The wavelength and the modulation SCLK control the duration of the entire wave through the following relationship:

Duration = Modulation SCLK / Wavelength

Each of the parameters has a finite length and therefore, the duration has maximum and minimum intervals. The modulation SCLK has a range of 1 Hz to 2.5 MHz and the Wavelength is limited from 2 points to 30,000 points. As a result, the duration can be programmed from 800 ns to 30,000 s.

If you do not care to control the wavelength and the SCLK, then you can leave the task for the 3D composer. In that case you must leave the Force Length and Force SCLK check boxes – unmarked. If you check the Force SCLK box, the wavelength will be modified automatically to match the selected duration. If you check the Force Length box, the modulation SCLK will be modified automatically to match the selected duration. Finally, if you check both the Force Length and the Force Modulation SCLK boxes, the duration of the 3D profile will be affected.

To modify wavelength or modulation SCLK, check the appropriate box, modify the value and click on the Apply button to force the selected value. Any successive changes that you make to the edit fields require that you click on the Apply button to accept the new value.

The Offset group controls DC offsets of the modulated waveform. Changing offset does not affect other parameters except the location of the waveform along the vertical axis.

The Clear Design button resets the 3D composer and the Reduce button closes the dialog box.

Vertical Controls

The *Vertical Controls* are used for profiling amplitude, frequency and phase. When you modify the fields in any of the controls, the associated graphical screen are automatically updated with the assigned values and display the profile as designed in the vertical control fields. The Vertical Controls are shown in Figure 4-61. You can start designing profiles only when one of the control fields is active. Control fields become active when you click on a control field.

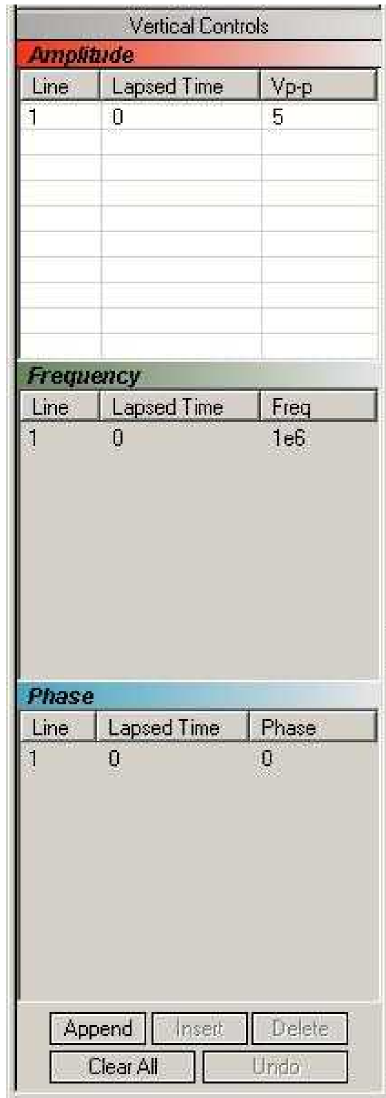

Figure 4-61, 3D Vertical Controls

Graphical Screens

The 3D *Waveform Graphs* are shown in Figure 4-62. You cannot change anything on the screens. However, anything that you design in the Vertical Controls fields will automatically be updated and displayed on the graphical screens.

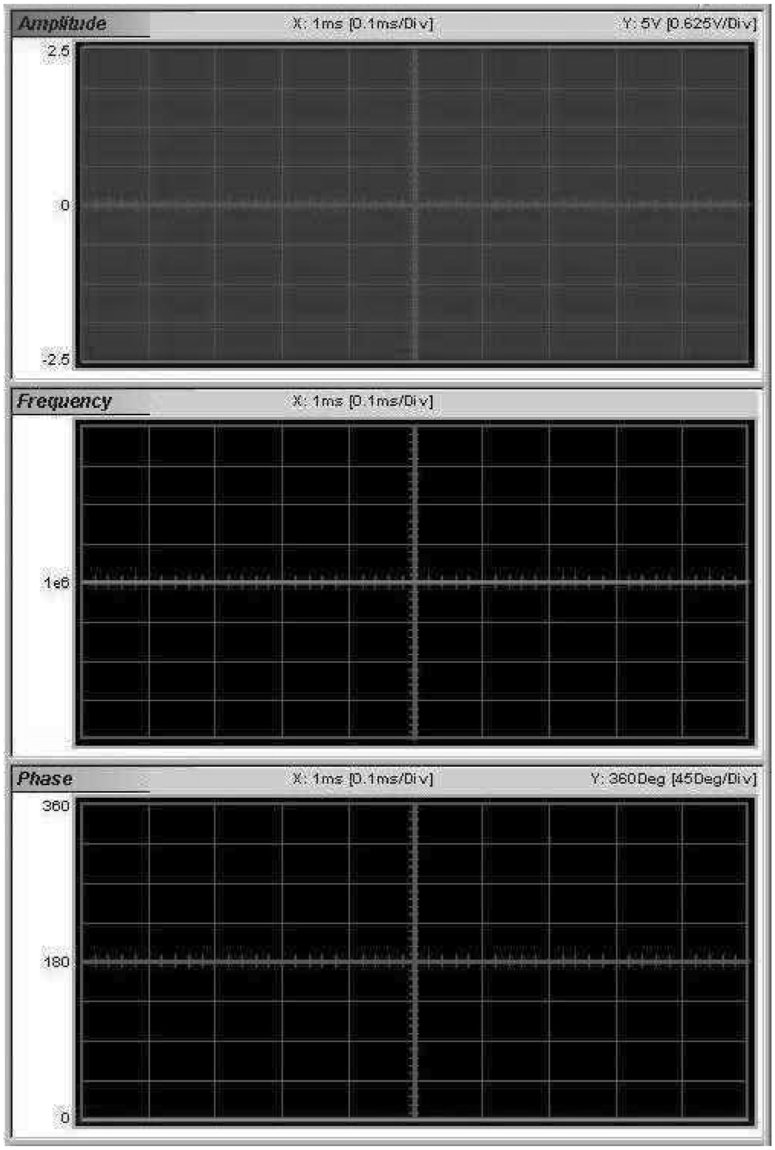

Figure 4-62, 3D Waveform Graphs

Designing 3D Profiles

3D profiles are designed in the Vertical Controls fields. Notice that there are three separate control fields: Amplitude, Frequency and Phase.

Always start the design from the Shared Horizontal Controls group. In the View group, remove profiles that you do not care to change. Click on the Parameters tab and set up the duration of the waveform. An example of a 3D profile (chirp, in this example) is shown in Figure 4-63. Profiles were designed for amplitude, frequency and phase. As you can see the duration of the waveform was selected to be 100 ms.

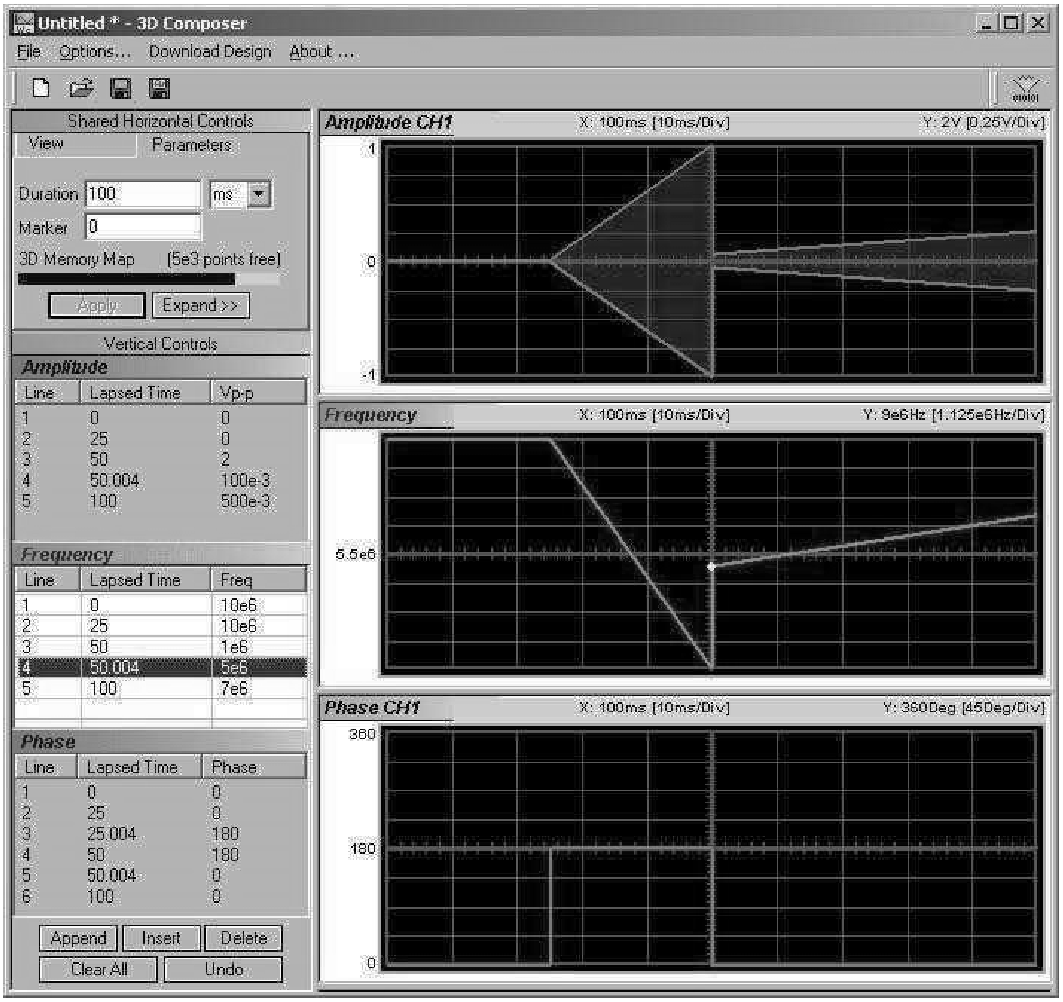

Figure 4-63, 3D Chirp Design Example

The Command Editor

The Command Editor is a tool for doing low-level programming of the 3172. Invoke the Command Editor from the System menu at the top of the screen. The Command Editor dialog box, as shown in Figure 4-64, will pop up. If you press the Download button, the function call in the Command field will be sent to the instrument.

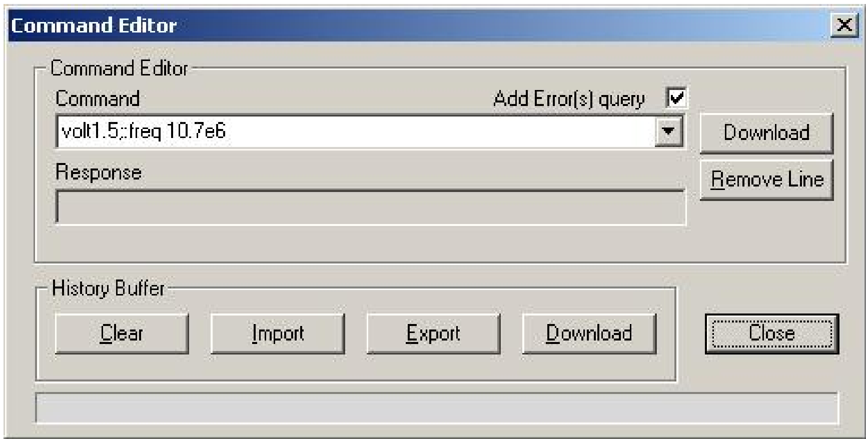

Figure 4-64, Command Editor

Low-level SCPI commands and queries can be sent directly to the 3172 from the **Command** field. Instrument responses to queries automatically appear in the **Response** field. The command editor is a useful troubleshooting tool. This way you can be sure of command syntax and functionality before you use it in your application. The complete list of 3172 SCPI commands is available in Chapter 5.

Logging SCPI Commands

The Log File is very useful for programmers that do not wish to spend a lot of time on manuals. When you use ArbConnection, every time you click on a button or change parameter, the command is logged in the same format as should be used in external applications. Figure 4-65 shows an example of a log file and a set of SCPI commands as resulted from some changes made on ArbConnection panels. You can set up the 3172 from ArbConnection to the desired configuration, log the commands in the log file and then copy and paste to your application without any modifications. Of course, this is true for simple commands that do not involve file download but, on the other hand, this is a great tool to get you started with SCPI programming.

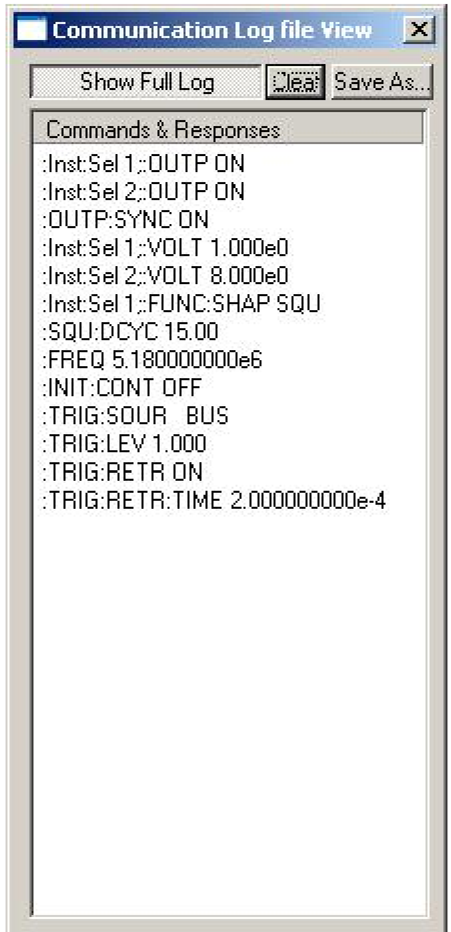

Figure 4-65, Log File Example

This page was intentionally left blank.

Chapter 5
Programming Reference

What's in this Chapter

This Chapter lists and describes the set of SCPI-compatible (Standard Commands for Programmable Instruments) remote commands used to operate the 3172. To provide familiar formatting for users who have previously used the SCPI reference documentation, the command descriptions are dealt with in a similar manner. In particular, each sub-system's documentation starts with a short description, followed by a table showing the complete set of commands in the sub-system; finally the effects of individual keywords and parameters are described. A complete listing of all commands used for programming the 3172 in 3171 LEGacy emulation mode is given in Table 5-1 and MODern mode in Table 5-2.

Introduction to SCPI

Commands to program the instrument over the GPIB are defined by the SCPI 1993.0 standard. The SCPI standard defines a common language protocol. It goes one step further than IEEE-STD-488.2 and defines a standard set of commands to control every programmable aspect of the instrument. It also defines the format of command parameters and the format of values returned by the instrument.

SCPI is an ASCII-based instrument command language designed for test and measurement instruments. SCPI commands are based on a hierarchical structure known as a tree system. In this system, associated commands are grouped together under a common node or root, thus forming subsystems.

Part of the OUTPut subsystem is shown below to illustrate the tree system:

```
:OUTPut
  :FILTer
     [:LPASs]                        {NONE|25M|50M|ALL}
        [:STATe] OFF | ON
```

OUTPut is the root keyword of the command; FILTer and STATe are second level keywords. LPASs is third level keyword. A colon (:) separates a command keyword from a lower level keyword.

Command Format

The format used to show commands in this manual is shown below:

```
FREQuency {<frequency>|MINimum|MAXimum}
```

The command syntax shows most commands (and some parameters) as a mixture of upper and lowercase letters. The uppercase letters indicate the abbreviated spelling for the command. For shorter program lines, send the abbreviated form. For better program readability, use the long form.

For example, in the above syntax statement, FREQ and FREQUENCY are both acceptable forms. Use upper or lowercase letters. Therefore, FREQ, FREQUENCY, freq, and Freq are all acceptable. Other forms such as FRE and FREQUEN will generate an error.

The above syntax statement shows the frequency parameter enclosed in triangular brackets. The brackets are not sent with the command string. A value for the frequency parameter (such as "FREQ 50e+6") must be specified.

Some parameters are enclosed in square brackets ([]). The brackets indicate that the parameter is optional and can be omitted. The brackets are not sent with the command string.

Command Separator

A colon (:) is used to separate a command keyword from a lower level keyword as shown below:

```
SOUR:FUNC:SHAP SIN
```

A semicolon (;) is used to separate commands within the same subsystem, and can also minimize typing. For example, sending the following command string:

```
TRIG:SLOP NEG;COUN 10;TIM 5e-3
```

is the same as sending the following three commands:

```
:TRIG:SLOP                                            NEG
:TRIG:COUN                                             10
:TRIG:TIM 5e-3
```

Use the colon and semicolon to link commands from different subsystems. For example, in the following command string, an error is generated if both the colon and the semicolon are not used.

```
OUTP:STATE ON;:TRIG:BURS ON
```

The MIN and MAX Parameters

Substitute MINimum or MAXimum in place of a parameter for some commands. For example, consider the following command:

```
FREQuency {<frequency>|MINimum|MAXimum}
```

Instead of selecting a specific frequency, substitute MIN to set the frequency to its minimum value or MAX to set the frequency to its maximum value.

Querying Parameter Setting

Query the current value of most parameters by adding a question mark (?) to the command. For example, the following command sets the output function to square:

```
SOUR:FUNC:SHAP SQR
```

Query the output function by executing:

```
SOUR:FUNC:SHAP?
```

Query Response Format

The response to a query depends on the format of the command. In general, a response to a query contains current values or settings of the generator. Commands that set values can be queried for their current value. Commands that set modes of operation can be queried for their current mode. IEEE-STD-488.2 common queries generate responses, which are common to all IEEE-STD-488.2 compatible instruments.

SCPI Command Terminator

A command string sent to the function generator must terminate with a <new line> character. The IEEE-STD-488 EOI message is a <new line> character. Command string termination always resets the current SCPI command path to the root level.

IEEE-STD-488.2 Common Commands

The IEEE-STD-488.2 standard defines a set of common commands that perform functions like reset, trigger and status operations. Common commands begin with an asterisk (*), are four to five characters in length, and may include one or more parameters. The command keyword is separated from the first parameter by a blank space. Use a semicolon (;) to separate multiple commands as shown below:

```
*RST; *STB?; *IDN?
```

SCPI Parameter Type

The SCPI language defines four different data formats to be used in program messages and response messages: numeric, discrete, Boolean, and arbitrary block.

Numeric Parameters

Commands that require numeric parameters will accept all commonly used decimal representations of numbers including optional signs, decimal points, and scientific notation. Special values for numeric parameters like MINimum and MAXimum are also accepted.

Engineering unit suffices with numeric parameters (e.g., MHz or kHz) can also be sent. If only specific numeric values are accepted, the function generator will ignore values, which are not allowed and will generate an error message. The following command is an example of a command that uses a numeric parameter:

```
VOLT:AMPL <amplitude>
```

Discrete Parameters

Discrete parameters are used to program settings that have a limited number of values (i.e., FIXed, USER and SEQuence). They have short and long form command keywords. Upper and lowercase letters can be mixed. Query responses always return the short form in all uppercase letters. The following command uses discrete parameters:

```
SOUR:FUNC:MODE {FIXed | USER | SEQuence}
```

Boolean Parameters

Boolean parameters represent a single binary condition that is either true or false. The generator accepts "OFF" or "0" for a false condition. The generator accepts "ON" or "1" for a true condition. The instrument always returns "0" or "1" when a boolean setting is queried. The following command uses a boolean parameter:

```
OUTP:FILT { OFF | ON }
```

The same command can also be written as follows:

```
OUTP:FILT {0 | 1 }
```

Arbitrary Block Parameters

Arbitrary block parameters are used for loading waveforms into the generator's memory. Depending on which option is installed, the 3172 can accept binary blocks up to 1M bytes. The following command uses an arbitrary block parameter that is loaded as binary data:

```
TRAC:DATA#564000<binary_block>
```

Binary Block Parameters

Binary block parameters are used for loading segment and sequence tables into the generator's memory. Information on the binary block parameters is given later in this manual.

SCPI Syntax and Styles

Where possible the syntax and styles used in this section follow those defined by the SCPI group of the IVI foundation. The commands on the following pages are broken into three columns; the Keyword, the Parameter Form, Default and HS command equivalent.

The Keyword column provides the name of the command. The actual command consists of one or more keywords since SCPI commands are based on a hierarchical structure, also known as the tree system. Square brackets ([ ]) are used to enclose a keyword that is optional when programming the command; that is, the 3172 will process the command to have the same effect whether the optional node is omitted by the programmer or not. Letter case in tables is used to differentiate between the accepted short form (upper case) and the long form (upper and lower case).

The Parameter Form column indicates the number and order of parameter in a command and their legal value. Parameter types are distinguished by enclosing the type in angle brackets (< >). If parameter form is enclosed by square brackets ([ ]) these are then optional (care must be taken to ensure that optional parameters are consistent with the intention of the associated keywords). The vertical bar (|) can be read as "or" and is used to separate alternative parameter options.

Legacy vs. Modern Command Set

The installed base of the legacy 3171 Arbitrary Waveform Generator & Dual Pulse Generator is large although, ultimately, it had to be discontinued because of parts obsolescence. While replacing such a popular instrument with modern technology presented minimal challenges, supporting the installed base with existing code and tested procedures created some compatibility challenges. The real challenge was to design a state-of-the-art product that has modern features but allows use of the 3172 at a level fully compatible with the legacy 3171.

The front panel of the 3172 looks similar to that of the 3171. So, if the 3172 is intended as a replacement in a legacy system, the 3171 legacy code is useful and, therefore, the instrument defaults to the legacy 3171-compatible mode. The SCPI commands that are shown in Tables 5-1 and 5-2 list the legacy 3171 commands set and marks the areas where the 3172 could not be made identical. Notice however, that you may still use the form:inst mod to be able to use the additional functionality that was built into the new 3172 design but expect that is you do just that, some of the legacy commands will not be compatible anymore.

CAUTION

Due to some differences in waveform resolution and size, waveforms cannot be shared across the legacy and modern options. Therefore, as a general rule of thumb, using the form:inst (leg | mod) command is not recommended before you read and fully understand the

consequences that may result in your application

In general the legacy 3172 mode modifies modern 3172 performance in the following major areas:

- Maximum sample clock frequency is reduced to 100 MS/s;
- Waveform interlace is set to 2;
- Minimum waveform length is set to 10; and
- Vertical resolution of arbitrary waveforms is reduced to 12 bits.

The 3172 can be pre-configured to emulate the 3171 legacy code and therefore, the instrument defaults to the legacy 3171-compatible mode with the above limitations built into the code. The non-emulation versions default to a different set of values to allow full performance of the generator; These commands are summarized in Table 5-1.

3172 Legacy Commands

The 3172 is a modern and updated version of the Model 3171 employing the latest technology and component improvements.

Although 100% backwards compatibility was the basis for the new 3172 design, some of the 3171 functionality is not supported. Table 5-1 lists all of the original 3171 SCPI commands and provides a check mark in the 3172 column in cases where the 3172 emulates the command. Expect unsupported commands to generate errors.

Table 1, 3172 SCPI Command Summary for 3171 Emulation

Keyword	Parameter Form	Default	PG1 n=1	PG2 n=2	ARB n=3	3172
[:SOURce<n>]						
:FUNCtion						
:SHAPe	DC SINusoid SQUare TRIangular RAMP PULSe USER DIGital	DC	x x	x x	x x x x x x x	✓
:POLarity	NORMal | INVerted	NORM			x	✓
:COUNt	<numeric value>	0			x	✓
:VOLTage						✓
:MODE	FIXed LIST	FIX	x	x	x x	✓ ✓
[:LEVel]						✓
[:IMMediate]						✓
[:AMPLitude]	<numeric value>	0.00	x	x	x	✓
:TRIGgered						✓
[:AMPLitude]	<numeric value>	0.00	x	x	x	✓
:OFFSet	<numeric value>	0.00	x	x	x	✓
:REFerence						✓
:STATe	ON | OFF	OFF			x	✓
:FREQuency						✓
:MODE	CW	CW	x	x	x	✓
[:CW]	<numeric value>	1e3			x	✓
PULSe						✓
:PERiod	<numeric value>	1.25e-8	x	x	x	✓
:WIDTh	<numeric value>	1e-8	x	x	x	✓
:COUNt	<numeric value>	0	x	x	x	✓
:POLarity	NORMal | INVerted	NORM	x	x	x	✓
:TRANsition						✓
[:LEADing]	<numeric value>	5e-9	x	x		✓
:TRAiling	<numeric value>	5e-9	x	x		✓
:STATe	OFF | ON	OFF	x	x		✓

Table 5-1, 3172 SCPI Command Summary for 3171 Emulation (continued)

Keyword	Parameter Form	Default	PG1 n=1	PG2 n=2	ARB n=3	3172
[:SOURce<n>]					x	✓
:LIST					x	✓
:VOLTage	<numeric value> {,<numeric value>}	No default value			x	✓
:REPeat	<numeric value> {,<numeric value>}	No default value			x	✓
:COUNt	<numeric value>	0			x	✓
:GENeration	DSEQuence	DSEQ			x	✓
:SRATe	<numeric value>	No default value			x	✓
:AM						✓
:SOURce	INTernal | EXTernal	INT			x	✓
:STATe	OFF | ON	OFF			x	✓
:ROSCillator					x	✓
:SOURce	INTernal | EXTernal	INT			x	✓
:OUTPut<n>			x	x	x	✓
:IMPedance	0 | 50 | 93	0	x	x	x	✓
[:STATe]	OFF | ON	OFF	x	x	x	✓
:TTLTrg<y>						✓
:SOURce	INT<1 to 8>	INT2	x	x	x	✓
:STATe	OFF | ON	OFF	x	x	x	✓
:FILTer					x	✓
[:LPASs]					x	✓
:FREQuency	2e6 | 2.2e6 | 22e6	22e6			x	✓
:STATe	OFF | ON	OFF			x	✓
:EXTernal			x	x		✓
:STATe	OFF | ON	OFF	x	x		✓
:TRIGger			x	x	x	✓
[:SEQuence]			x	x	x	✓
:SOURce	EXTernal | TTLTrg<y> | IMMediate	IMM	x	x	x	✓
:SLOPe	POSitive | NEGative	POS	x	x	x	✓
:DELay	<numeric value>	1e-8	x	x	x	✓
:GATE			x	x	x	✓
:SOURce	TTLTrg<y>	TTLT8	x	x	x	✓
:LEVel	HIGH | LOW	HIGH	x	x	x	✓
:DELay	<numeric value>	-2.01e153	x	x	x	
:INITiate<n>			x	x	x	✓
:IMMediate]		No default value	x	x	x	✓
:CALibration			x	x	x	✓
:ALL		No default value	x	x	x	
:DATA3						
:STARt						
ADDRess	#H<numeric hex value>	No default value			x	
:MEM3						
:DATA	#H<numeric hex value>	No default value			x	
:DIGital3					x	✓
:STIMulus					x	✓
:PATTern						
[:VALue]	<numeric value> {,<numeric value>}	No default value			x	✓
:REPeat	<numeric value> {,<numeric value>}	No default value			x	✓
:TIMing					x	✓
:CLOCk	<numeric value>	-6.87983e43			x	✓
:COUNt	<numeric value>	0			x	✓

Table 5-1, 3172 SCPI Command Summary for 3171 Emulation (continued)

Keyword	Parameter Form	Default	PG1 n=1	PG2 n=2	ARB n=3	3172
:SYSTem			x	x	x	✓
:ERROr?		0, No Error	x	x	x	✓
:VERSion?		1992.0	x	x	x	✓
:PRESet<n>		No default value	x	x	x	✓
:STATus					x	
:OPERation					x	
[:EVENt]?		0	x	x	x	
:CONDition?		Not implemented	x	x	x	
:ENABle	<nrf>	0	x	x	x	
:ENABle?		0	x	x	x	
:QUEStionable					x	
[:EVENt]?		0	x	x	x	
:CONDition?		Not implemented	x	x	x	
:ENABle	<nrf>	0	x	x	x	
:ENABle?		0	x	x	x	
:PRESet		No default value	x	x	x	

3172 Commands

Table 5-2 lists all of the 3172 SCPI commands that represent full functionality of the instrument. Note that there are some commands that pertain to W6 modules only and some only to P2 modules. Table 5-2 separates and associates the commands that control the different modules. The W6 designation implies that these commands apply to the arbitrary waveform generator and the P2 designation implies that the pulse generator can be programmed. When W6 and P2 are listed (W6, P2), both the arbitrary waveform generator and the pulse generator could be affected, depending on the selected output channel. 3172 in the association column implies that the command affects the entire system, regardless of whether W6 or P2 modules are installed. W6 SCPI commands are also valid for the W2 model.

Table 2, 3172 SCPI Command List Summary

Keyword	Parameter Form	Default	Association
Instrument Control Commands			
:INSTrument			
[:SELect]	1 | 2 | 3 | 4	1	W6, P2
:COUPle			
:MODE	MASTer | SLAVe	MAST	W6W6
:DELay	0 to 20	0	W6W6
:PATH	ADJacent | ECLT | LBUS	ADJ	W6W6
:SLAVe			
:DELete	<LAN_IP_address>		3172
:INSert	<3172>,<LAN_IP_address>		3172
:STATe	OFF | ON | 0 | 1	0	W6
:FORMat	MODern | LEGacy	MOD	3172
Run Mode Commands			
:INITiate			
[:IMMediately]			
:CONTinuous	OFF | ON | 0 | 1	1	W6, P2
:TRIGger			
[:IMMediate]			
:BURSt			
[:STATe]	OFF | ON | 0 | 1	0	W6, P2
:COUNt	1 to 1000000, W6; 1 to 65,536, P2	1	W6, P2
:DELay			
:STATe	OFF | ON | 0 | 1	0	W6, P2
:TIMe	100e-9 to 20, W6; 100e-9 to 1, P2	100e-9	W6, P2
:GATE			
:MODe	LEVel | TRANsition	LEV	W6
[:STATe]	OFF | ON | 0 | 1	0	W6, P2
:LEVel	-10 to 10	1.6	W6
:SOURce			
[:ADVance]	EXTernal | INTernal | TTLTrg<n> | ECLTrg1 | BUS | ADJcent	EXT	W6, P2
:SLOPe	POSitive | NEGative	POS	W6, P2
:RETRigger			
[:STATe]	OFF | ON | 0 | 1	0	W6
:TIMe	100e-9 to 20	100e-9	W6
:TIMer	1e-6 to 20	15e-6	W6

Table 5-2, Model 3172-W6 SCPI Commands List Summary (continued)

Keyword	Parameter Form	Default	Association
Output Control Commands			
:OUTPut		0	
:AMODulation	OFF | ON | 0 | 1	50	P2
:IMPedance	0 | 50 | 93	50	W6, P2
:FILTer			
[:LPASs]			W6
:FREQuency	2MHz | 25MHz | 60MHz | 120 MHz	25MHz	W6
[:STATe]	OFF | ON | 0 | 1	0	W6
:LOAD	50 to 1e6	50	W6
[:STATe]	OFF | ON | 0 | 1	0	W6, P2
:SYNC			
[:STATe]	OFF | ON | 0 | 1	0	W6, P2
:POSition			
[:POINt]	0 to 1e6-1	0	W6
:SOURce	BIT | LCOMplete | SSYNc | PULSe |ZEROcross	BIT	W6
:WIDTh	4 to 64	4	W6
:TRIGger			
:SOURce	BIT | LCOMPlete | INTernal | EXTernal	BIT	W6, P2
:TTLTrg	0 to 7	0	W6, P2
:SOURce	1 | 2 | 3 | 4	1	3172
[:STATe]	OFF | ON | 0 | 1	0,0	W6, P2
[:SOURce]			
:ROSCillator			
:SOURce	INTernal | EXTernal | CLK10 (EXT in W6 only)	INT	W6, P2
:FREQuency			
:SOURce	INTernal | ADJacent | RASTer | LBUS	INT	P2
:EXTernal?			W6
[:CW]	10e-3 to 30e6 | MINimum | MAXimum	1e6	W6
:RASTer	10e-6 to 200e6 | MINimum | MAXimum	1e6	W6
:SOURce	INTernal | EXTernal | ECLTrg0 | LBUS	INT	W6
:VOLTage			
[:LEVel]			
[:AMPLitude]	5e-3 to 20 | MINimum | MAXimum	1/2 range	W6, P2
:OFFSet	-10 to 10	Centered for range	W6, P2
:HILevel	-15.995 to 20	2.5	P2
:LOLevel	-16 to 19.995	-2.5	P2
:RANGe	SYMMetrical | POSitive | NEGative	SYMM	W6, P2
:PHASe			
[:OFFSet]	0 to 1e6-1	0	W6
:FUNCtion			
:MODE	FIXed | USER | SEQuence | MODulation | HALFcycle | COUNter	FIX	W6

Table 5-2, Model 3172 SCPI Commands List Summary (continued)

Keyword	Parameter Form	Default	Association
Standard Waveforms Commands			
[:SOURce]			
:SHAPe	SINusoid | TRIangle | SQUare | PULSe | RAMP | SINC | GAUSsian | EXPonential | NOISe | DC	SIN	W6
:SINusoid			
:PHASe	0 to 360	0	W6
:POWer	1 to 9	1	W6
:TRIangle			
:PHASe	0 to 360	0	W6
:POWer	1 to 9	1	W6
:SQUare			
:DCYCle	0 to 99.99	50	W6
:PULSe			
:DELay	0 to 99.999	10	W6
:WIDth	0 to 99.999	10	W6
:TRANsition			
[:LEADing]	0 to 99.999	10	W6
:TRAiling	0 to 99.999	10	W6
:RAMP			
:DELay	0 to 99.99	10	W6
:TRANsition			
[:LEADing]	0 to 99.99	10	W6
:TRAiling	0 to 99.99	10	W6
:SINC			
:NCYCle	4 to 100	10	W6
:GAUSsian			
:EXPonent	1 to 200	10	W6
:EXPonential			
:EXPonent	-100 to 100	-10	W6
:DC			
[:VOLTage]	-100 to 100	100	W6
Arbitrary Waveforms Commands			
:FORMat			
:BORDer	NORMal | SWAPped	NORM	W6
:TRACe			
[:DATA]	<data_array>		W6
:DEFine	<1 to 10k>,<16 to 1e6> (<segment_#>,<size>)	1	W6
:DELete			
[:NAME]	1 to 10k		W6
:ALL			W6
:DIGital		1	W6
:DATA			W6
:RATE	10e-6 to 200e6 | MINimum | MAXimum	1e6	W6
[STATe]	OFF | ON | 0 | 1	1	W6
:SELect	1 to 10k	1	W6
:SEGMent			
[:DATA]	<data_array>		W6

Table 5-2, Model 3172 SCPI Commands List Summary (continued)

Keyword	Parameter Form	Default	Association
Apply Commands			
[:SOURce]			
:APPLy	FREQ,AMPL,OFFS		W6
:SINusoid	FREQ,AMPL,OFFS,PHAS,POW		W6
:TRIangle	FREQ,AMPL,OFFS,PHAS,POW		W6
:SQUare	FREQ,AMPL,OFFS,DCY		W6
:PULSe	FREQ,AMPL,OFFS,DEL,WID,LEE,TRE		W6
:RAMP	FREQ,AMPL,OFFS,DEL,LEE,TRE		W6
:SINC	FREQ,AMPL,OFFS,CYC		W6
:GAUssian	FREQ,AMPL,OFFS,EXP		W6
:EXPonential	FREQ,AMPL,OFFS,EXP		W6
:DC	DC_AMPL		W6
:USER	SEG<n>,SCLK,AMPL,OFFS		W6
Sequence Commands			
[:SOURce]			W6
:SEQuence			
[:DATA]	<data_array>		W6
:ADVance	AUTOmatic | TRIGgered | STEP | MIX	AUTO	W6
:SELect	1 to 10	1	W6
:DEFine	<step>,<seg_#>,<repeat>,<adv_mode_x>,<sync_bit_x>		W6
:DELete			
[:NAME]	1 to 4096		W6
:ALL			W6
:SYNC			
[:TYPe]	LCOMplete | BIT	LCOM	W6
Modulated Waveforms Commands			
[:SOURce]			
:MODulation			W6
:TYPE	OFF | AM | FM | SWE | FSK | ASK | PSK | FHOPping | AHOPping | EXAM	OFF	W6
:CARRier			
[:FREQuency]	10 to 30e6	1e6	W6
:BASeline	CARRier | DC	CARR	W6

Table 5-2, Model 3172 SCPI Commands List Summary (continued)

Keyword	Parameter Form	Default	Association
Modulated Waveforms Commands (continued)			
[:SOURce]			
:AM			
:FUNCtion			
:SHAPe	SINusoid | TRIangle | SQUare | RAMP	SIN	W6
:INTernal			
:FREQuency	10e-3 to 1e6	100	W6
:DEPTh	0 to 100	50	W6
:EXEcute		(= MOD:TYPE AM)	W6
:FM			
:DEViation	10.0e-3 to 30e6	100e3	W6
:FUNCtion			
:SHAPe	SINusoid | TRIangle | SQUare | RAMP | ARB	SIN	W6
:FREQuency	10e-3 to 350e3	10e3	W6
:RASTer	1 to 2.5e6	1e6	W6
:MARKer			
[:FREQuency]	10e-3 to 30e6	1e6	W6
:DATA	<data_array>		W6
:SWEep			
:FREQuency			
[:STARt]	10 to 30e6	10e3	W6
:STOP	10 to 30e6	1e6	W6
:RASTer	10e-6 to 200e6 | MINimum | MAXimum	1e6	W6
:FUNCtion	SINusoidal | TRIangle(*) | SQUare(*)	SIN (*) Computed	W6
:TIME	1.4e-6 to 40.0	1e-3	W6
:DIRection	UP | DOWN	UP	W6
:SPACing	LINear | LOGarithmic	LIN	W6
:MARKer			
[:FREQuency]	10 to 30e6	505e3	W6
:FSK			W6
:FREQuency			
:SHIFted	10e-3 to 30e6	100e3	W6
:BAUD	1 to 10e6	10e3	W6
:MARKer	1 to 4000	1	W6
:DATA	<data_array>		W6

Table 5-2, Model 3172 SCPI Commands List Summary (continued)

Keyword	Parameter Form	Default	Association
Modulated Waveforms Commands (continued)			
[:SOURce]			
:ASK			
[:AMPLitude]			
[:STARt]	0 to 10	5	W6
:SHIFted	0 to 10	1	W6
:BAUD	1 to 2.5e6	10e3	W6
:MARKer	1 to 1000	1	W6
:DATA	<data_array>		W6
:PSK			
:PHASe			
[:STARt]	0 to 360	0	W6
:SHIFted	0 to 360	180	W6
:RATE	1 to 10e6	10e3	W6
:DATA	<data_array>		W6
:MARKer	1 to 4000	1	W6
:FHOPping			
:DWELl			
:MODe	FIXed | VARiable	FIX	W6
[:TIMe]	200e-9 to 20	200e-9	W6
:FIXed			
:DATA	<data_array>		W6
:VARiable			
:DATA	<data_array>		W6
:MARKer	1 to 1000	1	W6
:AHOPping			
:DWELl			
:MODe	FIXed | VARiable	FIX	W6
[:TIMe]	200e-9 to 20	200e-9	W6
:FIXed			
:DATA	<data_array>		W6
:VARiable			
:DATA	<data_array>		W6
:MARKer	1 to 5000	1	W6

Table 5-2, Model 3172 SCPI Commands List Summary (continued)

Keyword	Parameter Form	Default	Association
Pulse Waveforms Commands			
[:SOURce]			
:PULSe			
:MODe	NORMal | DELayed | DOUBle | HOLDdcycle | EWIDth	NORM	P2
:PERiod	20e-9 to 10	1e-3	P2
:WIDTh	7e-9 to 10	100e-6	P2
:DCYCle	1 to 99	50	P2
:DELay	0 to 10	0	P2
:DOUBle			
:DELay	0 to 10	200e-6	P2
:POLarity	NORMal | COMPlement | INVerted	NORM	P2
:TRANsition			
:STATe	FAST | LINear | SYMMetrical	FAST	P2
[:LEADing]	10e-9 to 5e-3	10e-6	P2
:TRAiling	10e-9 to 5e-3	10e-6	P2
Half Cycle Commands			
:HALFcycle			
:DELay	200e-9 to 20	1e-6	W6
:DCYCle	0 to 99.99	50	W6
:FREQuency	10e-3 to 1e6	1e6	W6
:PHASe	0 to 360	0	W6
:SHAPe	SINusoid | TRIangle | SQUare	SIN	W6
Counter Commands			
:COUNter			
:FUNCtion	FREQuency | PERiod | APERiod | PULSe | TOTalize	FREQ	W6
:DISPlay			
:MODE	NORMal | HOLD	NORM	W6
:GATE			
:TIME	100e-6 to 1	1	W6
:RESet			W6
:READ			W6
Power Amplifier Commands			
:OUTPut			
:AMPLifier	AUTO | HIGH | LOW	AUTO	A3
:POWer	OFF | ON | 0 | 1	0	A3

Table 5-2, Model 3172 SCPI Commands List Summary (continued)

Keyword	Parameter Form	Default	Association
LAN Configuration Commands			
:SYSTem			
:IP			
[:ADDRess]	<IP_address>		3172
:MASK	<mask>		3172
:GATeway	<gate_way>		3172
:BOOTp	OFF | ON | 0 | 1	0	3172
HOSTname:	<host_name>		3172
:KEEPalive			
:STATe	OFF | ON | 0 | 1	1	3172
:TIMEout	2 to 300	45	3172
:PROBes	2 to 10	2	3172
USB Mass Memory Commands			
:MMEMory			
:CATalog?			3172
:CDIRectory	<directory_name>		3172
:DELete	<file_name>		3172
:LOAD			
:TRACe	<label>,<file_name>		3172
System Commands			
:RESet			W6, P2
:SYSTem			
:ERRor?			W6, P2
:VERSion?			3172
:INFOrmation			
:CALibration?			3172
:MODel?			3172
:SERial?			3172
:SMEMory			
:MODe	READ | WRITe	WRIT	W6
[:STATe]	OFF | ON | 0 | 1	0	W6
:TEST			
[:ALL]?			3172
*CLS			3172
*ESE	1 to 255	1	3172
*OPC			3172
*RST			3172
*SRE	1 to 255	1	3172
*TRG			3172
*ESE?			3172
*ESR?			3172
*IDN?			3172
*OPC?			3172
*OPT?			3172
*STB?			3172
*TST?			3172

Instrument & Output Control Commands

This group is used to control the state, amplitude and offset settings of a channel, as well as the waveform mode. You can also synchronize multiple instruments and program phase offsets between adjacent channels. The output frequency and the reference source are also selected using commands from this group. Table 5-3 summarizes the Instrument and Output Control Commands. Factory defaults after *RST are shown in the Default column. Parameter range and low and high limits are listed, where applicable.

Table 3, Instrument & Output Control Command Summary

Keyword	Parameter Form	Default	Association
:INSTrument			
[:SELect]	1 | 2 | 3 | 4	1	W6,P2
:COUPle			
:MODE	MASTer | SLAVe	MAST	3172-W6W6 only
:DELay	0 to 20	0	3172-W6W6 only
:PATH	ADJacent | ECLT | LBUS	ADJ	3172-W6W6 only
:SLAVe			
:DELete	<LAN_IP_address>		3172
:INSert	<3172>,<LAN_IP_address>		3172
:STATe	OFF | ON | 0 | 1	0	W6
FORMat	MODern | LEGacy	MOD	3172
:OUTPut			
:AMODulation	OFF | ON | 0 | 1	0	W6
:IMPedance	0 | 50 | 93	50	W6,P2
:ECLTrg<n>	<n> = 0 to 1		W6,P2
[:STATe]	OFF | ON | 0 | 1	0,0	W6,P2
:FILTer			
[:LPASs]			
:FREQuency	2MHz | 25MHz | 60MHz | 120 MHz	25MHz	W6
[:STATe]	OFF | ON | 0 | 1	0	W6
:LOAD	50 to 1e6	50	W6
[:STATe]	OFF | ON | 0 | 1	0	W6,P2
:SYNC			
[:STATe]	OFF | ON | 0 | 1	0	W6
:POSition			
[:POINt]	0 to 1e6-1	0	W6
:SOURce	BIT | LCOMplete | SSYNc | PULSe | ZEROcross	BIT	W6
:WIDTh	4 to n-8	4	W6
:TRIGger			
:SOURce	BIT | LCOMPlete | INTernal | EXTernal	BIT	W6, P2
:TTLTrg	0 to 7	0	W6,P2
:SOURce	1 | 2 | 3 | 4	1	W6,P2
[:STATe]	OFF | ON | 0 | 1	0,0	W6,P2

Table 5-3, Instrument & Output Control Command Summary (continued)

Keyword	Parameter Form	Default	Association
[:SOURce]			
:ROSCillator			
:SOURce	INTernal | EXTernal | CLK10	INT	W6,P2
:FREQuency			
:EXTernal?			
:SOURce	INTernal | ADJacent | RASTer | LBUS	INT	P2
[:CW]	10e-3 to 30e6 | MINimum | MAXimum	1e6	W6
:RASTer	10e-6 to 200e6 | MINimum | MAXimum	1e6	W6
:SOURce	INTernal | EXTernal | ECLTrg0 | LBUS	INT	W6
:VOLTage			
[:LEVel]			
[:AMPLitude]	5e-3 to 20 | MINimum | MAXimum	1/2 range	W6,P2
:OFFSet	-10 to 10	Centered for range	W6,P2
:HILevel	-15.995 to 20	2.5	P2
:LOLevel	-16 to 19.995	-2.5	P2
:RANGe	SYMMetrical | POSitive | NEGative	SYMM	W6,P2
:FUNCtion			
:MODE	FIXed | USER | SEQuence | MODulation | HALFcycle | COUNter	FIX	W6

INSTrument {1|2|3|4}(?)

Description

This command sets the active instrument for future programming sequences. Subsequent commands affect the selected instrument only. The 3172 carrier can hold a combination of W6 and P2 cards. This command selects the active channel regardless of whether it is an arb or pulse channel. Refer to Figure 5-1 and then compare to what you have printed on your serial number label to determine the association between the selected channel number and what it will actually control on your instrument.

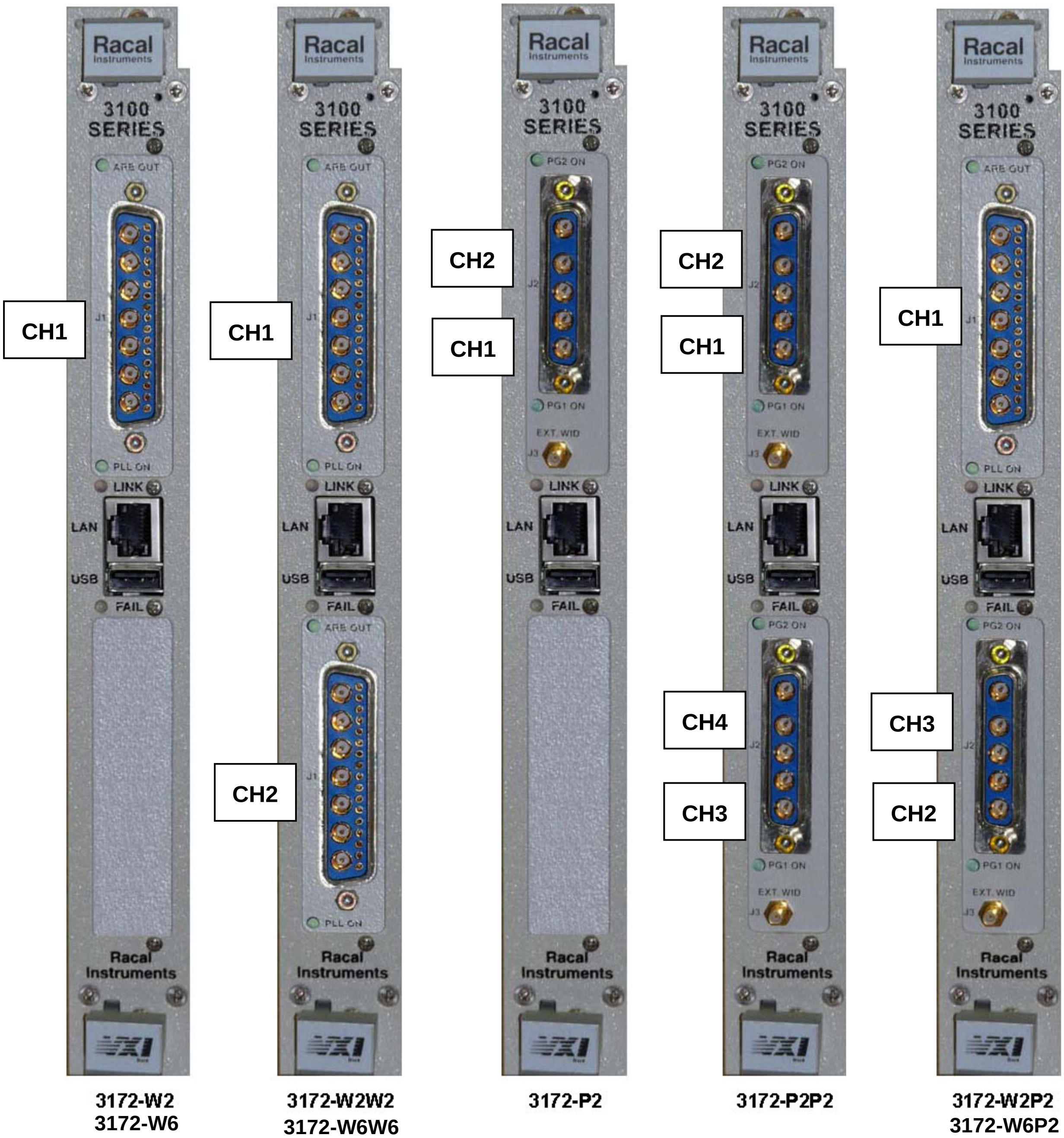

Figure 5-1. 3172 Instrument Channel Numbering Conventions for Various Configurations

Parameters

Range	Type	Default	Description
1-4	Discrete	1	Sets the active instrument for programming from remote.

Response

The 3172 returns 1, 2, 3 or 4 depending on the present active channel setting.

INSTrument:COUPle:MODE {MASTer|SLAVe}(?)

Description

This command assigns master or slave properties to the instrument. If the assignment is slave, most of the instrument operational functions will be controlled from the master instrument. Waveforms, amplitudes and offsets can be controlled individually for each slave unit.

Parameters

Name	Type	Default	Description
MASTer	Discrete	MAST	Programs a specific 3172 in a multi-instrument system as master instrument. Note that only one instrument can be designated as master while all other instruments may be programmed as slaves.
SLAVe	Discrete		Programs a 3172 in a multi-instrument system as a slave instrument. Note that multiple instruments can be designated as slaves but only one instrument can be designated as master.

Response

The 3172 returns MAST or SLAV depending on the current instrument coupled mode assignment.

INSTrument:COUPle:DELay <delay>(?)

Description

This command programs the delay time between the master and slave instruments. The waveform start on the slave units is delayed in reference to the master start point.

Parameters

Name	Range	Type	Default	Description
<delay>	0 to 20	Numeric	0	Sets the waveform start delay between instruments in units of seconds. Instrument 1 is always the master and reference channel. Instruments 2 to “n” are delayed in reference to instrument 1. Note that this parameter is operating in conjunction with the continuous run mode and only when multiple instruments are synchronized.

Response

The 3172 returns the present delay value in units of seconds.

INSTrument:COUPle:PATH {ADJacent|ECLT|LBUS}(?)

Description

This command selects the source path for multi-instrument synchronization.

Parameters

Name	Type	Default	Description
ADJacent	Discrete	ADJ	Selects the adjacent source path. Adjacent path is automatically configured between two instruments in the same carrier in the following configurations only: 3172-W6W6 and 3172-W6P2.
ECLT	Discrete		Selects the backplane ECLTrg as the synchronization path. In this case, the ecl trigger lines are turned on and connect between slots to provide the synchronization signals. Note that VXI backplane ECL trigger lines are limited to carry frequencies below 66 MHz.
LBUS	Discrete		Selects the backplane LBUS lines as the synchronization path. LBUS lines typically support 3172 operation through its entire frequency range. To use the LBUS, the master and slave instruments must be installed into a contiguous group of VXI chassis slots, with the master on the left. Also, the LBUS jumpers must be installed. For details, refer to the Local Bus Configuration section in Chapter 2.

Response

The 3172 returns ADJ, ECLT or LBUS depending on the current instrument couple path assignment.

INSTrument:COUPle:SLAVe:DELete <LAN_address>

Description

This command deletes a designated slave instrument from a synchronized multi-instruments system list. This command is associated with LAN operation only.

Parameters

Name	Type	Default	Description
<LAN_address>	String		Will remove a designated instrument, which is specified through its IP address, from a synchronized multi-instruments system list. Contact your computer administrator, if you are not sure how to specify a LAN address.

INSTrument:COUPle:SLAVe:INSert <3172>,<LAN_address>

Description

This command will add a designated slave instrument to a synchronized multi-instruments system list. This command is associated with LAN operation only.

Parameters

Name	Type	Default	Description
<3172>, <LAN_address>	String		Specifying the correct model number is crucial for correct assignment of the instrument designators, for selecting the correct instrument number for the INST:SEL command.

INSTrument:COUPle:STATe {OFF|ON|0|1}(?)

Description

This command turns the 3172 coupling state on and off.

Parameters

Range	Type	Default	Description
0-1	Discrete	0	Sets the coupling mode to on or off. Note that this command must be applied to the master instrument only. To select the master instrument use the INST:SEL 1 command.

Response

The 3172 returns 1 if the coupled state is on or 0 if the coupled state is off.

INSTrument:FORMat {MODern|LEGacy}(?)

Description

This command selects operation of the 3172 as a legacy replacement of the 3171 or operation as a modern instrument with all of the features that are described in the 3172-W6 specifications. Note that every time you toggle between modern and legacy modes, the instrument automatically resets all of its parameters and operating modes to the defaults that are associated with each mode.

Parameters

Range	Type	Default	Description
MODern	Discrete	MOD	This is the default selection for the 3172-W6 and all of the commands that are described in this chapter apply to this format.
LEGacy	Discrete		This selects the legacy format. If the instrument was ordered as a legacy replacement, it will be shipped with this format as default and there are no other actions that are required to convert this instrument to perform legacy functions. If the instrument was ordered without emulation of the 3171, this command transforms the 3172 modern instrument to behave and feel like the legacy 3171. When switching formats from modern to legacy and vice-versa, the output defaults automatically to the preset values that are associated with each format.

Response

The 3172 returns MOD or LEG depending on the current instrument format setting.

OUTPut:AMODulation {OFF|ON|0|1}(?)

Description

This command toggles on and off the external amplitude modulation input. When turned on, any signal, as specified and applicable for the modulation input, will amplitude modulate the output. The W2 and W6 configurations apply the AM modulation differently, with the W6 being compatible with the legacy 3171.

Parameters

Type	Default
Discrete	0

Response

The 3172 returns 1 if the input is turned on or 0 if the input is turned off.

OUTPut:IMPedance {0|50|93}(?)

Description

This command selects which source impedance is connected between the output amplifier and the output connector. Load impedances applied to the output connector will affect the output level except when the “0” impedance option is selected.

WARNING: Do not select 0 Ω mode and short circuit the output continuously as this could damage the output amplifier.

Parameters

Name	Type	Default	Description
0	Discrete		Selects 0 Ω as the source impedance
50	Discrete	50	Selects 50 Ω as the source impedance
93	Discrete		Selects 93 Ω as the source impedance

Response

The 3172 returns 0, 50, or 93 depending on the source impedance setting.

OUTPut:ECLTrg<n> {OFF|ON|0|1}(?)

Description

This command converts ECLTRG lines to outputs and places some signal on these lines to be used by other instruments in the chassis as synchronization signals. Although the original purpose of these lines was to be used as ECL triggers to other instruments, the 3172 is using these lines to synchronize clocks and start signals with other instruments in the chassis. The ECLTRG lines run in parallel from slot to slot so it is not important where the receiving module is placed inside the chassis, as long as the receiving instrument assigns the same lines as inputs. Only two of these trigger lines are used: ECLTRG0 and ECLTRG1.

Parameters

Name/Range	Type	Default	Description
<n>	Numeric (integer only)	0	Will specify the ECL trigger line that will be affected by this command. Only two lines are available in this case, 0 and 1.
0-1	Discrete	0	Will specify the state of the designated ECL trigger line either on or off.

Response

For ECLTRG0, the 3172 returns 0,1 if the output is on, or 0,0 if the output is off.

For ECLTRG1, the 3172 returns 1,1 if the output is on, or 1,0 if the output is off.

OUTPut:FILTer {2M|25MH|60M|120M}(?)

Description

This command selects which filter is connected to the 3172 output. Observe the following restrictions when you try to use this command:

Filter selection is not available when the instrument is set to output the standard sine waveform. In fact, the default waveform shape is sine. Therefore, filter selection will be available for use only after you select a different waveform, or change the output mode to use.

Parameters

Name	Type	Default	Description
2M	Discrete		Connects a 2MHz, Bessel type filter, to the output path
25M	Discrete		Connects a 25MHz, Bessel type filter, to the output path
60M	Discrete		Connects a 60MHz, Elliptic type filter, to the output path
120M	Discrete		Connects a 120MHz, Elliptic type filter, to the output path

Response

The 3172 returns NONE, 2M, 25M, 60M, or 120M depending on the type of filter presently connected to the output.

OUTPut:FILTer {OFF|ON|0|1}(?)

Description

This command toggles on and off filters that were selected with the OUTP:FILT command. Note that the filters are not accessible during standard sine waveform output. And therefore, if you intend to use filters, change the output waveform function or type first and then you’ll be allowed to apply filters as required.

Parameters

Range	Type	Default	Description

0-1	Discrete	0	Sets the output on and off

Response

The 3172 returns 1 if a filter is selected or 0 if a filter is de-selected.

OUTPut:LOAD <load>(?)

Description

This command specifies the load impedance that will be applied to the 3172 output.

Parameters

Name	Type	Default	Description
<load>	Numeric (integer only)	50	Will specify the load impedance that will be applied to the 3172 outputs in units of Ω. The default setting is 50 Ω. The range of load impedance is 50 Ω to 1 MΩ. Accurate setting of the load impedance is crucial for correct value of the amplitude level at the load.

Response

The 3172 returns an integer value depending on the present output load setting.

OUTPut {OFF|ON|0|1}(?)

Description

This command turns the 3172 output on and off. Note that for safety, the outputs always default to off, even if the last instrument setting before power down was on

Parameters

Range	Type	Default	Description
0-1	Discrete	0	Sets the output on and off

Response

The 3172 returns 1 if the output is on or 0 if the output is off.

OUTPut:SYNC {OFF|ON|0|1}(?)

Description

This command turns the 3172 SYNC output on and off. Note that for safety, the SYNC output always defaults to off, even if the last instrument setting before power down was on

Parameters

Range	Type	Default	Description
0-1	Discrete	0	Sets the SYNC output on and off

Response

The 3172 returns 1 if the SYNC output is on or 0 if the SYNC output is off.

OUTPut:SYNC:POSition <position>(?)

Description

This command programs the 3172 SYNC position. This command is active in arbitrary (USER) mode only.

Parameters

Name	Range	Type	Default	Description
<position>	0 to 1e6-1	Numeric (Integer only)	0	Sets the SYNC position in waveform points. The sync position can be programmed in increments of 4 points minimum.

Response

The 3172 returns the present SYNC position value

OUTPut:SYNC:SOURce {BIT|LCOM|SSYN|PULS|ZERO}(?)

Description

This command programs the condition that needs to be validated for the 3172 to generate the SYNC output.

Parameters

Name	Type	Default	Description
BIT	Discrete	BIT	Programs the bit validation. This type is the normal sync output for the standard and arbitrary waveforms. The bit output will revert to marker output when the instrument is programmed to generate modulated waveforms. The width of the bit pulse is always four sample clock cycles and cannot be changed, however, the position of the bit pulse in reference to the waveform cycle can be changed using the outp:sync:pos command. In case the bit is too narrow for the application, you may use the outp:sync:sour puls option where you can modify both the position and the width of the sync signal
LCOMplete	Discrete		Programs the loop complete validation. This type is the normal sync output for the sequenced and counted burst waveforms. The output will revert automatically to LCOM when the instrument is programmed to generate one of the above waveforms. The LCOM pulse starts at the beginning of the sequence and ends at the end of the sequence. You may change the start position using the outp:sync:pos command. In this case the LCOM pulse will start at the new position but will always end at the end of the sequence
SSYNc	Discrete		Programs the synchronized validation. This type is similar to the bit option except it is useful in triggered or

		gated modes where the ±1 clock jitter between the trigger and the output signal is eliminated through a special circuit that synchronizes the sync output to the triggered signal
PULSe	Discrete	Programs the pulse validation. This type is similar to the bit option except that when this option is selected the width of the sync pulse can be programmed in increments of 4 points from a minimum of 4 point to a maximum of segment length minus 8 waveform points. The position of the pulse is programmed using the outp:sync:pos command and its width is programmed using the outp:sync:wid command.
ZEROcross	Discrete	Programs the zero crossing validation. This type is a special mode where the sync signal remains low as long as the output waveform level is negative (below 0 V) but changes to high when the output level becomes positive

Response

The 3172 returns BIT, LCOM, SSYN, PULS, or ZERO depending on the selected SYNC validation option.

OUTPut:SYNC:WIDTh <width>(?)

Description

This command programs the 3172 SYNC position. This command is active in arbitrary (USER) mode only.

Parameters

Name	Range	Type	Default	Description
<width>	4 to 60	Numeric (Integer only)	4	Sets the SYNC width in waveform points. The sync width can be programmed in increments of 4 points minimum.

Response

The 3172 returns the present SYNC width value.

OUTPut:TRIGger:SOURce {BIT|LCOMplete|INTernal|EXTernal}(?)

Description

The TTLTRG signals, when enabled and placed on the backplane, can be asserted with signals coming from a number of sources. Use this command to assign the source for the active TTLTRG line.

Parameters

Name	Type	Default	Description
BIT	Discrete	BIT	Generates a trigger signal at a designated point on the

		waveform. The trigger position is programmed using the outp:sync:pos command. The same command sets the position of the trigger output and the position of the SYNC output.
LCOMplete	Discrete	Generates a single trigger signal in sequence mode only once when the active segment appears for the first time.
INTernal	Discrete	This type selects the internal generator as the source.
EXTernal	Discrete	This type selects the external trigger input as the trigger source. An external signal must be connected to the TRIG IN connector for this mode to operate correctly.

Response

The 3172 returns BIT, LCOM, INT, or EXT depending on the present 3172 trigger source setting.

OUTPut:TRIGger:SOURce {P1|P2|SYNC1|SYNC2}(?)

Description

The TTLTRG signals, when enabled and placed on the backplane, can be asserted with signals coming from a number of sources. Use this command to assign the source for the active TTLTRG line. Use this command with p2 modules only.

Parameters

Name	Type	Default	Description
P1	Discrete	P1	Generates a trigger signal at a designated point on the waveform. The source of the trigger signal is the P1 channel.
P2	Discrete		Generates a trigger signal at a designated point on the waveform. The source of the trigger signal is the P2 channel.
SYNC1	Discrete		Generates a trigger signal at a designated point on the waveform. The source of the trigger signal is the sync output of the P1 channel.
SYNC2	Discrete		Generates a trigger signal at a designated point on the waveform. The source of the trigger signal is the sync output of the P2 channel.

Response

The 3172 returns P1, P2, SYNC1, or SYNC2 depending on the present 3172-P2 trigger source setting.

OUTPut:TTLTrg:SOURCe {1|2|3|4}(?)

Description

The SYNC signal can be placed on one of the backplane TTLTRG lines. Use this command to select the module that will source the trigger lines. For example, in a 3172-W6P2 configured instrument, “1” designates the arbitrary generator module and “2” and “3” designate the two pulse modules; Likewise, in a 3172P2P2 configured instruments, 1 to 4 designate the channels from top to bottom.

Parameters

Range	Type	Default	Description
1 to 4	Integer	1	Designates a specific 3172 module that will output the sync signal onto the backplane trigger outputs. The signals that are generated on the trigger lines can come from a number of sources, described in the outp:trig:sour command before.

Response

The 3172 returns 1, 2, 3 or 4 depending on the currently selected source module.

OUTPut:TTLTrg<n> <OFF|ON|0|1>(?)

Description

The TTLTRG lines can be used to transmit and receive trigger signals between the 3172 and other VXIbus modules. Use this command to transmit signals on the backplane trigger lines.

Parameters

Name	Range	Type	Default	Description
<n>	0-7,0-1	Integer	1	Designates a specific backplane trigger line as an output. The signals that are placed on these lines can come from a number of sources, described in the outp:trig:sour command before.

Response

The 3172 returns n,0 when a specific backplane trigger line is off or n,1 when a specific backplane trigger line is turned on. n can range from 0 to 7.

ROSCillator:SOURce {INTernal|EXTernal|CLK10}(?)

Description

This command selects the reference source for the sample clock generator.

Parameters

Name	Type	Default	Description
INTernal	Discrete	INT	Selects an internal source. The internal source is a 1ppm TCXO
EXTernal	Discrete		Activates the external reference input. An external reference must be connected to the 3172 for it to continue normal operation
CLK10	Discrete		Selects the backplane CLK10 source. The CLK10 clock

is routed in parallel to all backplane connectors and, therefore, all of the modules that are installed in the chassis can use the same clock source. This is particularly useful for synchronization purposes.

Response

The 3172 returns INT, EXT, or CLK10 depending on the present 3172 reference clock source setting.

FREQuency:EXTernal?

Description

This command queries the frequency at the trigger input. The same trigger is also used for phase locking to an external signal. This command returns the frequency value of the external signal only when the instrument is set to PLL mode.

Response

The 3172 measures and returns the present frequency value. If no signal is applied to the trigger input, the response will be 0. The returned value will be in standard scientific format (for example: 100mHz would be returned as 100e-3 – positive numbers are unsigned).

FREQuency:SOURce {INTernal|ADJacent}(?)

Description

This command is used for selecting the source of the clock generator for the P2 channels. If left with its default option, each of the pulse channels is fed from a separate and independent clock generator source. Use this command if you wish to synchronize the outputs of two pulse channels to each other or to the sample clock generator from the arbitrary waveform generator.

Parameters

Name	Type	Default	Description
INTernal	Discrete	INT	Each of the pulse channels has a separate clock generator. With this option, each channel can have a unique repetition rate, independent of the other channels in the 3172 carrier.
ADJacent	Discrete		Disables the clock in channel 2. The clock generator from channel 1 is fed to channel 2. In this case, the two pulse channels share a single clock and hence are synchronized to each other.

Response

The 3172 returns INT, EXT, or CLK10 depending on the present 3172 reference clock source setting.

FREQuency {<freq>|MINimum|MAXimum}(?)

Description

This command modifies the frequency of the standard waveforms in units of hertz (Hz). It has no effect on arbitrary waveforms.

Parameters

Name	Range	Type	Default	Description
<freq>	10e-3 to 30e6	Numeric	1e6	Sets the frequency of the standard waveform in units of Hz. The frequency command can be used with resolutions up to 14 digits. The accuracy of the instrument however, can only be tested to this accuracy using an external reference that provides the necessary accuracy and stability
<MINimum>		Discrete		Sets the frequency of the standard waveform to the lowest possible frequency (10e-3).
<MAXimum>		Discrete		Sets the frequency of the standard waveform to the highest possible frequency (30e6).

Response

The 3172 returns the present frequency value. The returned value will be in standard scientific format (for example: 100 mHz would be returned as 100e-3 – positive numbers are unsigned).

FREQuency:RASTer {<sclk>|MINimum|MAXimum}(?)

Description

This command modifies the sample clock frequency of the arbitrary waveform in units of samples per second (S/s). It has no effect on standard waveforms.

Parameters

Name	Range	Type	Default	Description
<sclk>	10e-6 to 200e6	Numeric	1e6	Sets the sample clock frequency of the arbitrary and sequenced waveform in units of S/s. The sample clock command can be programmed with resolutions up to 14 digits however, the accuracy can be tested to this accuracy using an external reference that provides the necessary accuracy and stability
<MINimum>		Discrete		Sets the sample clock frequency to the lowest possible frequency (10e-6).
<MAXimum>		Discrete		Sets the frequency of the standard waveform to the highest possible frequency (200e6).

Response

The 3172 returns the present sample clock frequency value. The returned value will be in standard scientific format (for example: 100 MHz would be returned as 100e6 – positive numbers are unsigned).

FREQuency:RASTer:SOURce {INTernal|EXTernal|ECLTrg0|LBUS<n>}(?)

Description

This command selects the source of the sample clock generator. This command affects both the standard, arbitrary and sequenced waveforms. The ECLTrg0 and the LBUS<n> sources are useful for applications requiring synchronization between adjacent modules where one is sourcing the clock through this line and one is using the clock on this line as the sample clock source.

Parameters

Name	Type	Default	Description
INTernal	Discrete	INT	Selects an internal source.
EXTernal	Discrete		Activates the external sample clock input. An external reference must be connected to the 3172, in the range of the internal source, for it to continue normal operation. Observe the input level limitations provided in Appendix A before connecting an external signal.
ECLTrg0	Discrete		Activates the backplane ECLTrg0 as the source for the sample clock input. Signal must be generated from another module in the chassis on this line otherwise the 3172 will not operate correctly. Observe the input level and limitations before connecting an external signal to this line.
LBUS<0-7>	Discrete		Activates the backplane LBUS0 to LBUS7 as the source for the sample clock input. The signal must be generated from an adjacent module in the chassis, acting as the master. Also, the master and slave modules must be configured for LBUS operation. For details, refer to the Local Bus Configuration section in Chapter 2.

Response

The 3172 returns INT, EXT, ECLT0, or LBUS<n> depending on the current sample clock source setting.

VOLTage {<ampl>|MINimum|MAXimum}(?)

Description

This command programs the peak to peak amplitude of the output waveform. The amplitude is calibrated when the source impedance is 50Ω.

Parameters

Name	Range	Type	Default	Description
<ampl>	5e-3 to 20e0	Numeric	10	Sets the amplitude of the output waveform in units of volts. Amplitude setting is always peak to peak. Offset and amplitude settings are independent providing that the offset + amplitude value does not exceed the specified window.
<MINimum>		Discrete		Sets the amplitude to the lowest possible level (5 mV).

MAXimum>	Discrete	Sets the amplitude to the highest possible level (20 V or 22 V depending on the range).

Response

The 3172 returns the present amplitude value. The returned value will be in standard scientific format (for example: 100 mV would be returned as 100e-3 – positive numbers are unsigned).

VOLTage:OFFSet <offs>(?)

Description

This command programs the amplitude offset of the output waveform. The offset is calibrated when the source impedance is 50Ω.

Parameters

Name	Range	Type	Default	Description
<offs>	-11 to 11	Numeric	0	Sets the offset of the output waveform in units of volts. Offset and amplitude settings are independent providing that the offset + amplitude do not exceed the specified window.

Response

The 3172 returns the present offset value. The returned value will be in standard scientific format (for example: 100 mV would be returned as 100e-3 – positive numbers are unsigned).

VOLTage:HILevel <high_level>(?)

Description

This command programs the amplitude high level value. This value must be programmed in conjunction with the amplitude low level value. Use this command to replace the amplitude/offset commands. The amplitude high level must be followed by the low level command.

Parameters

Name	Range	Type	Default	Description
<high_level>	-19.995 to 20	Numeric	2.5	Sets the amplitude high level value. Simultaneous programming of the amplitude low level value is mandatory.

Response

The 3172 returns the present high level value. The returned value will be in standard scientific format (for example: 100 mV would be returned as 100e-3 – positive numbers are unsigned).

VOLTage:LOLevel <low_level>(?)

Description

This command programs the amplitude high level value. This value must be programmed in conjunction with the amplitude low level value. Use this command to replace the amplitude/offset commands. The amplitude high level must be followed by the low level command.

Parameters

Name	Range	Type	Default	Description
<low_level>	-16 to 19.995	Numeric	-2.5	Sets the amplitude low level value. Simultaneous programming of the amplitude high level value is mandatory.

Response

The 3172 returns the present low level value. The returned value will be in standard scientific format (for example: 100 mV would be returned as 100e-3 – positive numbers are unsigned).

VOLTage:RANGe {SYMMetrical|POSitive|NEGative}(?)

Description

This command defines the range of the amplitude window. The window is selectable from symmetrical, where the amplitude is set between -11 V and 11 V, positive, where the amplitude is set between 0 V and 20 V and negative, where the amplitude can be programmed from 0 V to -16 V. Note that every time you modify the range, the output automatically defaults to the preset value that has been selected for each range.

Parameters

Name	Type	Default	Description
SYMMetrical	Discrete	SYMM	Selects the symmetrical window where amplitudes can be programmed between the -11 V to +11 V rails.
POSitive	Discrete		Selects the positive window where amplitudes can be programmed between the 0 V to +20 V rails.
NEGative	Discrete		Selects the negative window where amplitudes can be programmed between the 0 V to -16 V rails.

Response

The 3172 returns SYMM, POS, or NEG depending on the present 3172 output range setting.

PHASe:OFFSet <phase_offs>(?)

Description

This command affects a slave instrument only when synchronized to another module in the chassis. It programs the start phase offset in reference to an adjacent master module. Phase offset resolution when using this command is 1 point.

Parameters

Name	Range	Type	Default	Description
<phase_offs>	0 to 1e6-1	Numeric	0	Sets the phase offset in reference to a master

	(Integer only)		instrument. Slave instruments trail the master instrument edge.

Response

The 3172 returns the present phase offset value.

FUNCtion:MODE {FIXed|USER|SEQuence|MODulation|HALFcycle|COUNter}(?)

Description

This command defines the type of waveform that will be available at the output connector. It also selects one of the auxiliary functions from: counter/timer, digital pulse generator, and half cycle waveform generator.

Parameters

Name	Type	Default	Description
FIXed	Discrete	FIX	Selects the standard waveform shapes. There is an array of waveforms that is built into the program. You can find these waveform shapes in the standard waveforms section.
USER	Discrete		Selects the arbitrary waveform shapes. Arbitrary waveforms must be loaded to the 3172 memory before they can be replayed. You can find information on arbitrary waveforms in the appropriate sections in this manual.
SEQuenced	Discrete		Selects the sequenced waveform output. To generate a sequence, you must first download waveform coordinates to different segments and then build a sequence table to generate a complex waveform that is using these segments.
MODulated	Discrete		Selects the modulated waveforms. There is an array of built-in modulation schemes. However, you can also build custom modulation schemes using the arbitrary function.
HALFcycle	Discrete		Selects the half cycle function.
COUNter	Discrete		Selects the counter/timer auxiliary function. Note that when you select this function, all waveform generation of the 3172 is disabled and the 3172 is transformed into a stand-alone counter/timer.

Response

The 3172 returns FIX, USER, SEQ, MOD, HALF, or COUN depending on the present 3172 output function mode setting.

Run Mode Commands

The Run Mode Commands group is used to synchronize device actions with external events. These commands control the trigger modes of the waveform generator. The instrument can be placed in Triggered, Gated or Burst mode. Trigger source is selectable from an external, backplane trigger lines, an internal trigger generator that has asynchronous, free-running programmable intervals and software commands. It also has a built-in internal re-trigger generator that provides accurate and self-repeating control from waveform end to waveform start. Optional nodes were omitted from these commands. The Run Mode settings affect all waveform shapes equally except when using the modulated waveforms. In the case of modulated waveform, the output idles on either the carrier waveform or on a DC level until stimulated to output a modulation cycle or burst of cycles. Additional information on the run mode options and how the generator behaves in the various run mode options is given in Chapter 3. Table 5-4 summarizes the run mode commands. Factory defaults after *RST are shown in the default column. Parameter low and high limits are given where applicable.

Table 5-4, Run Mode Commands

Keyword	Parameter Form	Default	Association
:INITiate			
[:IMMediately]			
:CONTinuous	OFF | ON | 0 | 1	1	W6, P2
:TRIGger			
[:IMMediate]			
:BURSt			
[:STATe]	OFF | ON | 0 | 1	0	W6, P2
:COUNt	1 to 1000000, W6; 1 to 65,536, P2	1	W6, P2
:DELay			
:STATe	OFF | ON | 0 | 1	0	W6, P2
:TIMe	100e-9 to 20, W6; 100e-9 to 7, P2	100e-9	W6, P2
:GATE			
:MODe	LEVel | TRANsition	LEV	W6
[:STATe]	OFF | ON | 0 | 1	0	W6, P2
:LEVel	-10 to 10	1.6	W6
:SOURce			
[:ADVance]	EXTernal | INTernal | TTLTrg<n> | ECLTrg1 | BUS | ADJcent	EXT	W6, P2
:SLOPe	POSitive | NEGative	POS	W6, P2
:RETRigger			
[:STATe]	OFF | ON | 0 | 1	0	W6
:TIMe	100e-9 to 20	100e-9	W6
:TIMer	1e-6 to 20	15e-6	W6

INITiate:CONTinuous {1|0|ON|OFF}(?)

Description

This command sets the output in continuous operation and interrupted operation. The run mode commands will affect the 3172 only after it will be set to interrupted operation.

Parameters

Range	Type	Default	Description
1-0	Discrete	1	“1” selects the continuous run mode. “0” selects the interrupted run mode. While in this switch option, you can program the 3172 to operate in triggered, gated, or counted burst run modes.

Response

The 3172 returns 1 or 0 depending on the selected option.

TRIGger

Description

Use this command to trigger the 3172 from a remote computer. You may also use the common command *trg which has the same effect. This command will affect the 3172 after you program the instrument to operate in an interrupted run mode (init:cont 0) and only when you select the trigger source to be BUS.

Response

The 3172 will respond to a remote trig command depending on the selected mode and function.

TRIGger:BURSt {OFF|ON|0|1}(?)

Description

This command will toggle the counted burst run mode on and off. This command will affect the 3172 only after it will be set to INIT:CONT 0.

Parameters

Range	Type	Default	Description
0-1	Discrete	0	“1” enables the counted burst run mode. “0” turns the burst run mode off. Burst count is programmable using the TRIG:BURS:COUN command.

Response

The 3172 returns 0, or 1 depending on the selected option.

TRIGger:BURSt:COUNt <burst>(?)

Description

This function sets the number of cycles when the Burst Mode is on. Use the init:cont off;:trig:burs on commands to select the Burst Mode.

Parameters

Name	**Range**	**Type**	**Default**	**Description**
<burst>	1 to 1M	Numeric (integer only)	1	Programs the burst count. Note that <burst> for P2 is limited to 65,536 counts.

Response

The 3172 returns the present burst count value.

TRIGger:DELay {OFF|ON|0|1}(?)

Description

Use this command to turn on and off the delayed trigger function. The trigger delay time command will affect the generator only after it has been programmed to operate in interrupted run mode. Modify the 3172 to interrupted run mode using the init:cont off command. The trig:del 0 command duplicates this action.

Range	**Type**	**Default**	**Description**
0-1	Discrete	0	Turns the delayed trigger mode on and off.

Response

The 3172 returns 0, or 1 depending on the selected option.

TRIGger:DELay:TIMe <time>(?)

Description

The trigger delay time parameter defines the time that will elapse from a valid trigger signal to the initiation of the first output waveform. Trigger delay can be turned off and on using the trig:del <0|1> command. The trigger delay time command will affect the generator only after it has been programmed to operate in an interrupted run mode. Set the 3172 to be in an interrupted run mode using the init:cont off command.

Parameters

Name	**Range**	**Type**	**Default**	**Description**
<time>	100e-9 to 20	Numeric	100e-9	Programs the trigger delay time. Programming resolution is 20 ns across the range. Note that <time> for P2 is limited to 1 second.

Response

The 3172 returns the present trigger delay time value.

TRIGger:GATE:MODE {LEVel|TRANsition}(?)

Description

This command selects if the 3172 will gate between transitions or on level changes. The trig:slop command defines the polarity of both the transitions and the level. Select the source for the gating signal from the front panel TRIG IN connector or from one of the backplane trigger lines. However, note that if you want to control the trigger level threshold, you can only do it if you will be using the front panel input. This command will affect the 3172 only after it will be set to INIT:CONT OFF.

Parameters

Name	Type	Default	Description
LEVel	Discrete	LEV	Selects the mode where level change at the trigger input causes the output to turn the gate on and off.
TRANsition	Discrete		Selects transitions as the gating signals. First transition turns the gate on and second turns the gate off.

Response

The 3172 returns LEV, or TRAN depending on the selected option.

TRIGger:GATE {OFF|ON|0|1}(?)

Description

This command will toggle the gate run mode on and off. This command will affect the 3172 only after it is set to INIT:CONT 0 mode.

Parameters

Range	Type	Default	Description
0-1	Discrete	0	Turns the gate run mode off and on.

Response

The 3172 returns 0, or 1 depending on the selected option.

TRIGger:LEVel <level>(?)

Description

The trigger level command sets the threshold level at the trigger input connector. The trigger level command will affect the generator only after it has been programmed to operate in interrupted run mode. Modify the 3172 to interrupted run mode using the init:cont off command.

Parameters

Name	Range	Type	Default	Description
<level>	-10 to 10	Numeric	1.6	Programs the trigger level. The value affects the front panel input only. Note that this parameter is not available for the P2 module.

Response

The 3172 returns the present trigger level value.

TRIGger:SOURce:ADVance {EXTernal|INTernal|TTLTrg<n>|ECLT1|BUS}(?)

Description

This selects the source from where the 3172 will be stimulated to generate waveforms. The source advance command will affect the generator only after it has been programmed to operate in interrupted run mode. Modify the 3172 to interrupted run mode using the init:cont off command.

Parameters

Name	Type	Default	Description
EXTernal	Discrete	EXT	Selects the front panel TRIG IN input as the trigger source.
INTernal	Discrete		Activates the built in internal trigger generator. BUS and external trigger are ignored. The period of the internal trigger is programmable and can be used to replace an external trigger source.
TTLTrg<0 to 7>	Discrete		Selects the backplane trigger lines as the source for the trigger input. To avoid hardware conflicts, make sure that no more than one instrument is programmed to output trigger signals on any specific TTLT line.
ECLTrg1	Discrete		Selects the backplane ECL trigger line number 1 as the source for the trigger input.
BUS	Discrete		Selects the remote controller as the trigger source. Only software commands are accepted; Backplane and front panel signals are ignored
ADJacent	Discrete		When an active P2 channel is placed in triggered run mode, the clock generator of the other P2 channel feeds its trigger input. This feature is available in the P2 module only.

Response

The 3172 returns EXT, INT, TTLT<n>, BUS, or ADJ depending on the selected trigger source advance setting.

TRIGger:SLOPe {POSitive|NEGative}(?)

Description

The trigger slope command selects the sensitive edge of the trigger signal that is applied to the selected trigger source. The 3172 can be made sensitive to either the positive or negative transitions. Positive going transitions will trigger the generator when the POS option is selected. Negative transitions will trigger the generator when the NEG option is selected. In Gated mode, two transitions in the same direction are required to gate on and off the output. The trigger slope command will affect the generator only after it has been programmed to operate in interrupted run mode. Set the 3172 to be in an interrupted run mode using the init:cont off command.

Parameters

Name	Type	Default	Description
POSitive	Discrete	POS	Selects the positive going edge. When the level option

		is selected for the gate input, a positive level (above the trigger level setting) opens the gate.
NEGative	Discrete	Selects the negative going edge. When the level option is selected for the gate input, a negative level (below the trigger level setting) opens the gate.

Response

The 3172 returns POS, or NEG depending on the selected trigger slope setting.

RETRigger {OFF|ON|0|1}(?)

Description

This parameter turns on and off the re-trigger function. The re-trigger mode causes the 3172 to self-trigger at the end of the triggered signal. This has a completely different functionality than the internal trigger generator in the sense that once the instrument has been placed in internal trigger mode, the output generates continuous waveforms that are triggered by an internal trigger generator. The intervals of the internal trigger generator are measured from start waveform to start waveform. When the 3172 is prepared to operate in re-trigger mode, the output is waiting for an external or remote trigger signal; Once triggered, the instrument self-triggers automatically but this time, the intervals of the re-trigger generator are measured from the end of the waveform to the start of the next waveform. Re-trigger can be initiated from any of the selected advance options. The re-trigger command will affect the generator only after it has been programmed to operate in interrupted run mode. Set the 3172 to be in an interrupted run mode using the init:cont off command.

Parameters

Name	Type	Default	Description
0-1	Discrete	0	Turns the re-trigger mode on and off. Note that this parameter is not available for the P2 module.

Response

The 3172 returns 0 or 1 depending on the selected option.

RETRigger:TIMe <time>(?)

Description

This parameter specifies the amount of time that will elapse between the end of the delivery of the waveform cycle and the beginning of the next waveform cycle. Re-trigger can be initiated from any of the selected advance options. The re-trigger intervals are measured from waveform end to waveform start. The re-trigger command will affect the generator only after it has been programmed to operate in interrupted run mode. Modify the 3172 to interrupted run mode using the init:cont off command.

Parameters

Name	Range	Type	Default	Description
<time>	100e-9 to 20	Numeric	100e-9	Programs the re-trigger period. Programming resolution is 20 ns across the range. Note that this parameter is not available for the P2 module.

Response

The 3172 returns the present re-trigger period value.

TRIGger:TIMer <timer>(?)

Description

This parameter specifies the period of the internal trigger generator. This value is associated with the internal trigger run mode only and has no effect on other trigger modes. The internal trigger generator is a free-running oscillator, asynchronous with the frequency of the output waveform. The timer intervals are measured from waveform start to waveform start. Note the difference from the re-trigger mode where there the intervals are measured from waveform end to waveform start.

Parameters

Name	Range	Type	Default	Description
<time>	1e-6 to 20	Numeric	15e-6	Programs the internal trigger generator period. Note that this parameter is not available for the P2 module.

Response

The 3172 returns the present internal trigger period value.

Standard Waveforms Control Commands

This group is used to control the standard waveforms and their respective parameters. There is an array of standard waveforms that could be used without the need to download waveform coordinates to the instrument. You can also modify the parameters for each waveform to a shape suitable for your application.

Table 5-5 summarizes the standard waveforms control commands. Factory defaults after *RST are shown in the Default column. Parameter range and low and high limits are listed, where applicable.

Table 5, Standard Waveform Control Command Summary

Keyword	Parameter Form	Default	Association
[:SOURce]			
:SHAPe	SINusoid | TRIangle | SQUare | PULSe | RAMP | SINC | GAUSsian | EXPonential | NOISe | DC	SIN	W6
:SINusoid			
:PHASe	0 to 360	0	W6
:POWer	1 to 9	1	W6
:TRIangle			
:PHASe	0 to 360	0	W6
:POWer	1 to 9	1	W6
:SQUare			
:DCYCle	0 to 99.99	50	W6
:PULSe			
:DELay	0 to 99.999	10	W6
:WIDth	0 to 99.999	10	W6
:TRANsition			
[:LEADing]	0 to 99.999	10	W6
:TRAiling	0 to 99.999	10	W6
:RAMP			
:DELay	0 to 99.99	10	W6
:TRANsition			
[:LEADing]	0 to 99.99	10	W6
:TRAiling	0 to 99.99	10	W6
:SINC			
:NCYCle	4 to 100	10	W6
:GAUSsian			
:EXPonent	1 to 200	10	W6
:EXPonential			
:EXPonent	-100 to 100	-10	W6
:DC			
[:VOLTage]	-100 to 100	100	W6

FUNCtion:SHAPe {SINusoid|TRIangle|SQUare|PULSe|RAMP|SINC|GAUSsian|EXPonential|DC|NOISe}(?)

Description

This command defines the type of waveform that will be available at the output connector.

Parameters

Name	Type	Default	Description
SINusoid	Discrete	SIN	Selects the built-in sine waveform.
TRIangle	Discrete		Selects the built-in triangular waveform.
SQUare	Discrete		Selects the built-in square waveform.
SPULse	Discrete		Selects the built-in pulse waveform.
RAMP	Discrete		Selects the built-in ramp waveform.
SINC	Discrete		Selects the built-in sinc waveform.
EXPonential	Discrete		Selects the built-in exponential waveform.
GAUSsian	Discrete		Selects the built-in Gaussian waveform.
DC	Discrete		Selects the built-in DC waveform.
NOISe	Discrete		Selects the built-in noise waveform.

Response

The 3172 returns SIN, TRI, SQU, SPUL, RAMP, SINC, GAUS, EXP, DC, or NOIS depending on the present 3172 setting

SINusoid:PHASe <phase>(?)

Description

This command programs the start phase for the standard sine waveform. This command has no effect on arbitrary or modulated waveforms.

Parameters

Name	Range	Type	Default	Description
<phase>	0 to 360	Numeric	0	Programs the start phase parameter in units of degrees. Sine phase resolution is 0.1° limited however at high frequencies (above approximately 500 kHz), depending on the number of waveform points that are used to create the sine shape.

Response

The 3172 returns the present start phase value.

SINusoid:POWer <power>(?)

Description

This command programs power for the sine^x waveform. This command has no effect on arbitrary or modulated waveforms.

Parameters

Name	Range	Type	Default	Description
<power>	1 to 9	Numeric	1	Programs the power coefficient parameter for the sine waveform. The coefficient will have an effect up to approximately 500 kHz, depending on the number of waveform points that are used to create the sine shape.

Response

The 3172 returns the present power value.

TRIangle:PHASe <phase>(?)

Description

This command programs the start phase for the standard triangular waveform. This command has no effect on arbitrary or modulated waveforms.

Parameters

Name	Range	Type	Default	Description
<phase>	0 to 360	Numeric	0	Programs the start phase parameter in units of degrees. Triangle phase resolution is 0.1° limited however at high frequencies (above approximately 500 kHz), depending on the number of waveform points that are used to create the shape.

Response

The 3172 returns the present start phase value.

TRIangle:POWer <power>(?)

Description

This command programs power for the triangle^x waveform. This command has no effect on arbitrary or modulated waveforms.

Parameters

Name	Range	Type	Default	Description
<power>	1 to 9	Numeric	1	Programs the power coefficient parameter for the triangular waveform. The coefficient will have an effect up to approximately 500 kHz, depending on the number of waveform points that are used to create the shape.

Response

The 3172 returns the present power value.

SQUare:DCYCle <duty_cycle>(?)

Description

This command programs duty cycle of the standard square waveform. This command has no effect on arbitrary waveforms.

Parameters

Name	Range	Type	Default	Description
<duty_cycle>	0 to 99.99	Numeric	50	Programs the square wave duty cycle parameter in units of percent

Response

The 3172 returns the present duty cycle value.

PULSe:DELay <delay>(?)

Description

This command programs delay of the standard pulse waveform. This command has no effect on arbitrary waveforms.

Parameters

Name	Range	Type	Default	Description
<delay>	0 to 99.999	Numeric	10	Programs the pulse delay parameter in units of percent

Response

The 3172 returns the present pulse delay value.

PULSe:WIDTh <pulse_width>(?)

Description

This command programs pulse high portion of the standard pulse waveform. This command has no effect on arbitrary waveforms.

Parameters

Name	Range	Type	Default	Description
<pulse_width>	0 to 99.999	Numeric	10	Programs the pulse width parameter in units of percent

Response

The 3172 returns the present width value.

PULSe:TRANsition <rise>(?)

Description

This command programs pulse transition from low to high of the standard pulse waveform. This command has no effect on arbitrary waveforms.

Parameters

Name	Range	Type	Default	Description
<rise>	0 to 99.999	Numeric	10	Programs the pulse rise time parameter in units of percent

Response

The 3172 returns the present rise time value

PULSe:TRANsition:TRAiling <fall>(?)

Description

This command programs pulse transition from high to low of the standard pulse waveform. This command has no effect on arbitrary waveforms.

Parameters

Name	Range	Type	Default	Description
<fall>	0 to 99.999	Numeric	10	Programs the pulse fall time parameter in units of percent

Response

The 3172 returns the present fall time value.

RAMP:DELay <delay>(?)

Parameters

Name	Range	Type	Default	Description
<delay>	0 to 99.99	Numeric	10	Programs the ramp delay parameter in units of percent

Response

The 3172 returns the present ramp delay value.

Ramp:TRANsition <rise>(?)

Description

This command programs ramp transition from low to high of the standard ramp waveform. This command has no effect on arbitrary waveforms.

Parameters

Name	Range	Type	Default	Description
<rise>	0 to 99.99	Numeric	60	Programs the pulse rise time parameter in units of percent

Response

The 3172 returns the present rise time value

RAMP:TRANsition:TRAiling <fall>(?)

Description

This command programs ramp transition from high to low of the standard ramp waveform. This command has no effect on arbitrary waveforms.

Parameters

Name	Range	Type	Default	Description
<fall>	0 to 99.99	Numeric	30	Programs the ramp fall time parameter in units of percent

Response

The 3172 returns the present fall time value.

SINC:NCYCle <N_cycles>(?)

Description

This command programs the number of “0-crossings” of the standard SINC pulse waveform. This command has no effect on arbitrary waveforms.

Parameters

Name	Range	Type	Default	Description
<N_cycle>	4 to 100	Numeric (Integer only)	10	Programs the number of zero-crossings

Response

The 3172 returns the present number of zero-crossings specified.

GAUSsian:EXPonent <exp>(?)

Description

This command programs the exponent for the standard Gaussian pulse waveform. This command has no effect on arbitrary waveforms.

Parameters

Name	Range	Type	Default	Description
<exp>	1 to 200	Numeric	10	Programs the exponent parameter

Response

The 3172 returns the present exponent value.

EXPonential:EXPonent <exp>(?)

Description

This command programs the exponent for the standard exponential waveform. This command has no effect on arbitrary waveforms.

Parameters

Name	Range	Type	Default	Description
<exp>	-100 to 100	Numeric	-10	Programs the exponent parameter

Response

The 3172 returns the present exponent value.

DC <voltage>(?)

Description

This command programs the voltage level for the dc function. The peak to peak amplitude value is programmed using the volt<float> and this parameter programs the output level in units of percent relative to the programmed peak to peak amplitude level.

Parameters

Name	Range	Type	Default	Description
<voltage>	-100 to 100	Numeric	100	Programs level of the DC function in units of percent

Response

The 3172 returns the present DC voltage value.

Arbitrary Waveforms Control Commands (W6 Module Only)

This group is used to control the arbitrary waveforms and their respective parameters. This will allow you to create segments and download waveforms. Using these commands, you can also define segment size and delete some or all unwanted waveforms from your memory. Use the commands in this group to turn the digital output on and off and to download data to the digital pattern buffer.

Table 5-6 summarizes the arbitrary waveform control commands. Factory defaults after *RST are shown in the Default column. Parameter range and low and high limits are listed, where applicable.

Generating Arbitrary Waveforms

Arbitrary waveforms are generated from digital data points, which are stored in a dedicated waveform memory. Each data point has a vertical resolution of 16 bits (65536 points), i.e., each sample is placed on the vertical axis with a precision of 1/65536. The 3172 has 1M waveform memory capacity as standard.

Each horizontal point has a unique address - the first being 00000 and the last depends on the memory option. In cases where smaller waveform lengths are required, the waveform memory can be divided into smaller segments.

When the instrument is programmed to output arbitrary waveforms, the clock samples the data points (one at a time) from address 0 to the last address. The rate at which each sample is replayed is defined by the sample clock rate parameter.

Unlike the built-in standard waveforms, arbitrary waveforms must first be loaded into the instrument's memory. Correct memory management is required for best utilization of the arbitrary memory. An explanation of how to manage the arbitrary waveform memory is given in the following paragraphs.

Arbitrary Memory Management

The arbitrary memory in comprised of a finite length of words. The maximum size arbitrary waveform that can be loaded into memory depends on the option that is installed in your instrument. The various options are listed in Chapter 1 of this manual. If you purchased the 3172 with its basic configuration, you should expect to have 1 Meg words to load waveforms.

Waveforms are created using small sections of the arbitrary memory. The memory can be partitioned into smaller segments (up to 16k) and different waveforms can be loaded into each segment, each having a unique length. Minimum segment size is 16 points. Information on how to partition the memory, define segment length and download waveform data to the 3172 is given in the following paragraphs.

Table 6, Arbitrary Waveform Command Summary

Keyword	Parameter Form	Default	Association
:FORMat			
:BORDer	NORMal | SWAPped	NORM	W6
:TRACe			
[:DATA]	<data_array>		W6
:DEFine	<1 to 10k>,<16 to 1e6> (<segment_#>,<size>)	1	W6
:DELete			
[:NAME]	1 to 10k		W6
:ALL			
:DIGital		1	W6
:DATA			
:RATE	10e-6 to 200e6 | MINimum | MAXimum	1e6	W6
[STATe]	OFF | ON | 0 | 1	1	W6
:SELect	1 to 10k	1	W6
:SEGMent			
[:DATA]	<data_array>		W6

FORMat:BORDer {NORMal|SWAPped}(?)

Description

Binary data is sent to the instrument in byte-high byte-low order. For convenience, programmers can write their code in reverse order but have to let the instrument know that the data is reversed. In case the code stores the data in byte-low byte-high order, use this form:bord swap command to reverse the byte order.

Name	Type	Default	Description
NORMal	Discrete	NORM	Binary data will be sent in byte-high byte-low order
SWAPped	Discrete		Binary data will be sent in byte-low byte-high order

Response

The 3172 returns NORM or SWAP depending on the present format setting.

TRACe #<header><binary_block>

Description

This command will download waveform data to the 3172 memory. Waveform data is loaded to the 3172 using high-speed binary transfer. A special command is defined by IEEE-STD-488.2 for this purpose. High-speed binary transfer allows any 8-bit bytes (including extended ASCII code) to be transmitted in a message. This command is particularly useful for sending large quantities of data. As an example, the next command will download to the generator an arbitrary block of data of 1024 points

```
TRACe#42048<binary_block>
```

This command causes the transfer of 2048 bytes of data (1024 waveform points) into the active memory segment. The <header> is interpreted this way:

- The ASCII "#" ($23) designates the start of the binary data block.
- "4" designates the number of digits that follow.
- "2048" is the even number of bytes to follow.

The generator accepts binary data as 16-bit integers, which are sent in two-byte words. Therefore, the total number of bytes is always twice the number of data points in the waveform. For example, 20000 bytes are required to download a waveform with 10000 points. The IEEE-STD-488.2 definition of Definite Length Arbitrary Block Data format is demonstrated in Figure 5-2.

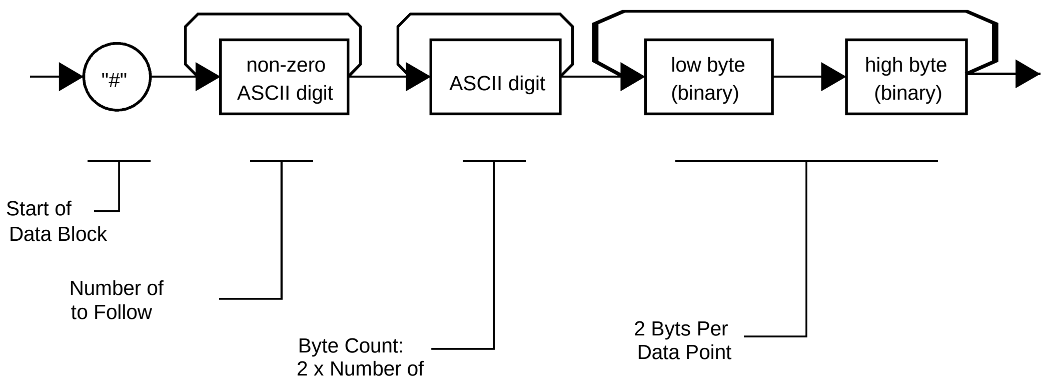

Figure 5-2, Definite Length Arbitrary Block Data Format

Transfer of definite length arbitrary block data must terminate with the EOI bit set. This way, carriage-return (CR – 0dH) and line feed (LF – 0aH) characters can be used as waveform data points and will not cause unexpected termination of the arbitrary block data.

- <binary_block> Represents waveform data.

The waveform data is made of 16-bit words. However, programmers may choose to prepare the data in two bytes and arrange to download these two bytes in a sequence. Figure 5-3 shows a waveform word that is acceptable for the 3172. There are a number of points you should be aware of before you start preparing the data:

1. Waveform data points have 16-bit values - 0x0000 to 0xFFFF
2. Data point range is 0 to 65,535 decimal for the 3172. 0x0000 correspond to -8 V and 0xFFFF corresponds to +8V
3. 3172 data point data point 65,535 corresponds to full-scale amplitude setting. Point 32768 corresponds to 0 V amplitude setting

Figure 5-3 shows how to initially prepare the 16-bit word for a waveform data point.

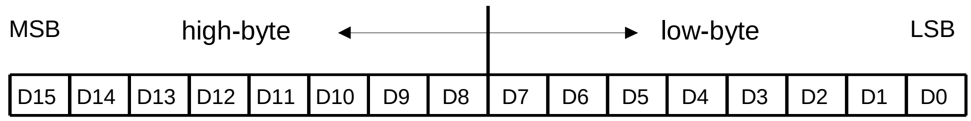

Figure 5-3, 3172-W6 16-bit Waveform Data Point Representation

Parameters

Name	Type	Description
<header>	Discrete	Contains information on the size of the binary block that contains waveform coordinates.
<binary_block>	Binary	Block of binary data that contains waveform data points (vertical coordinates), as explained above.

TRACe:DEFine <segment_# >,<length>

Description

Use this command to attach size to a specific memory segment. The final size of the arbitrary memory is 1M points. The memory can be partitioned to smaller segments, up to 10k segments. This function allows definition of segment size. Total length of memory segments cannot exceed the size of the waveform memory.

NOTE

The 3172 operates in interlaced mode where four memory cells generate one byte of data. Therefore, segment size can be programmed in numbers evenly divisible by four only. For example, 2096 bytes is an acceptable length for a binary block. 2002 is not a multiple of 4, therefore the generator will generate an error message if this segment length is used.

Parameters

Name	Range	Type	Default	Description
<segment_#>	1 to 10k	Numeric (integer only)	1	Selects the segment number of which will be programmed using this command
<length>	16 to n	Numeric (integer only)		Programs the size of the selected segment. Minimum segment length is 16 points, the maximum is limited by the total amount of installed memory.

TRACe:DELete <segment_number>

Description

This command will delete a segment. The memory space that is being freed will be available for new waveforms as long as the new waveform will be equal or smaller in size to the deleted segment. If the deleted segment is the last segment, then the size of another waveform written to the same segment is not limited. For example, let consider two segments, the first being a 1000-point waveform and the second with 100 points. If you delete segment 1, you can reprogram another waveform to segment 1 with size to 1000 points. If you reprogram segment 1 with 1004 points, the instrument will generate an error and will not accept this waveform. On the other hand, if you delete segment 2, which was the last segment you programmed, then you can reprogram this segment with waveforms having length limited only by the size of the entire memory space.

Parameters

Name	Range	Type	Default	Description
<segment_ number>	1 to 10k	Numeric (integer only)	1	Selects the segment number of which will be deleted

TRACe:DELete:ALL

Description

This command will delete all segments and will clear the entire waveform memory. This command is particularly important in case you want to de-fragment the entire waveform memory and start building your waveform segments from scratch.

TIP

The TRAC:DEL:ALL command does not re-write the memory so, whatever waveforms were downloaded to the memory are still there for recovery. The TRAC:DEL:ALL command removes all stop bits and clears the segment table. You can recover memory segments by using the TRAC:DEF command. You can also use this technique to resize, or combine waveform segments.

TRACe:SELect <segment_number>

Description

This command selects the active waveform segment for the output. By selecting the active segment you are performing two function:

1. Successive :TRAC commands will affect the selected segment

2. The SYNC output will be assigned to the selected segment. This behavior is especially important for sequence operation, where multiple segments form a large sequence. In this case, you can synchronize external devices exactly to the segment of interest

Parameters

Name	Range	Type	Default	Description
<segment_ number>	1 to 10k	Numeric (integer only)	1	Selects the active segment number

Response

The 3172 returns the active segment number.

SEGment #<header><binary_block>

Description

This command will partition the waveform memory to smaller segments and will speed up memory segmentation. The idea is that waveform segments can be built as one long waveform and then just use this command to split the waveform to the appropriate memory segments. In this way, there is no need to define and download waveforms to individual segments.

Using this command, segment table data is loaded to the 3172 using high-speed binary transfer in a similar way to downloading waveform data with the trace command. High-speed binary transfer allows any 8-bit bytes (including extended ASCII code) to be transmitted in a message. This command is particularly useful for large number of segment. As an example, the next command will generate three segments with 12 bytes of data that contains segment size information.

```
SEGment#212<binary_block>
```

This command causes the transfer of 12 bytes of data (3 segments) into the segment table buffer. The <header> is interpreted this way:

- The ASCII "#" ($23) designates the start of the binary data block.
- "2" designates the number of digits that follow.
- "12" is the number of bytes to follow. This number must divide by 4.

The generator accepts binary data as 32-bit integers, which are sent in two-byte words. Therefore, the total number of bytes is always 4 times the number of segments. For example, 36 bytes are required to download 9 segments to the segment table. The IEEE-STD-488.2 definition of Definite Length Arbitrary Block Data format is demonstrated in Figure 5-2. The transfer of definite length arbitrary block data must terminate with the EOI bit set. This way, carriage-return (CR – 0dH) and line feed (LF – 0aH) characters can be used as segment table data points and will not cause unexpected termination of the arbitrary block data.

The segment table data is made of 32-bit words however, the GPIB link has 8 data lines and accepts 8-bit words only. Therefore, the data has to be prepared as 32-bit words and rearranged as six 8-bit words before it can be used by the 3172 as segment table data. Figure 5-4 shows how to prepare the 32-bit work for the segment start address and size. There are a number of points you should be aware of before you start preparing the data:

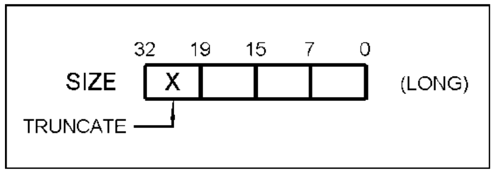

Figure 5-4, Segment Address and Size Example

1. Each channel has its own segment table buffer. Therefore, make sure you selected the correct active channel (with the INST:SEL command) before you download segment table data to the generator
2. Minimum number of segments is 1; maximum number of segments is 16k.
3. Maximum segment size depends on your installed option. With the basic 3172 you can program maximum 1M in one segment.
4. Segment table data has 32-bit values of which are used for segment size. Therefore, Data for each segment must have 4 bytes
5. The number of bytes in a complete segment table must divide by 6. The 3172 has no control over data sent to its segment table during data transfer. Therefore, wrong data and/or incorrect number of bytes will cause erroneous memory partition

Parameters

Name	**Type**	**Description**
<binary_block>	Binary	Block of binary data that contains information on the segment table.

The Apply Control Commands (W6 Module Only)

The apply commands combine popular commands into a string that contains all controls for a specific function. For example, to program a sine waveform that has certain frequency, amplitude and offset, you have to use five different commands:

func:mode fix

func:shap sin

freq <freq>

ampl <ampl>

offs <offs>

Alternately, you can select the sine function and immediately assign all of the required parameters when using the apply command. The five lines above are replaced by a simple line as follows:

Appl:sin <freq>,<ampl>, <offs>

There are certain rules you must follow when using the apply commands; These are summarized below.

Using the Apply Commands

The apply commands provide a high level method of programming pre-defined standard and arbitrary waveforms. Selection can be made for function, frequency, amplitude, offset and other parameters which are associated with the selected function. For example, the following statement outputs a 2 Vp-p square wave at 1 MHz with a 0 V offset and 10% duty cycle:

appl:squ 1e6, 2, 0, 10

It is not necessary to enter every parameter with the APPLy command. If only the frequency and offset need to be changed, omit the other parameters while keeping the commas. The other parameters are set to the power-up default values:

appl:squ 10e6,,1

Alternatively, if just the first parameters need to be changed, omit the commas. The other parameters are set to the power-up default values:

appl:squ 4e6,2

Queries can also be made on all parameters associated with a standard function using the appl:<function_shape>? query. For example, if the instrument was programmed using the above appl:squ command, query the square wave parameters using the following query:

appl:squ?

Table 5-7 lists the complete set of apply commands, followed by a description of each command separately.

Table 7, Apply Control Commands

Keyword	Parameter Form	Default	Association
[:SOURce]			
:APPLy	FREQ,AMPL,OFFS		W6
:SINusoid	FREQ,AMPL,OFFS,PHAS,POW		W6
:TRIangle	FREQ,AMPL,OFFS,PHAS,POW		W6
:SQUare	FREQ,AMPL,OFFS,DCY		W6
:PULSe	FREQ,AMPL,OFFS,DEL,WID,LEE,TRE		W6
:RAMP	FREQ,AMPL,OFFS,DEL,LEE,TRE		W6
:SINC	FREQ,AMPL,OFFS,CYC		W6
:GAUssian	FREQ,AMPL,OFFS,EXP		W6
:EXPonential	FREQ,AMPL,OFFS,EXP		W6
:DC	DC_AMPL		W6
:USER	SEG<n>,SCLK,AMPL,OFFS		W6

APPLy <freq>,<ampl>,<offs>(?)

Description

This command changes the waveform function to standard and programs the frequency, amplitude and offset for the selected standard waveform. This command affects the output regardless of the current output function. For example, if you generate FM, the 3172 will stop generating FM, will revert to a standard waveform and will update the values of the frequency, amplitude and offset, as specified by this command.

Parameters

Name	Range	Type	Default	Description
<freq>	10e-3 to 30e6	Numeric	1e6	Programs the frequency of the standard waveform in units of Hz.
<ampl>	10e-3 to 20	Numeric	5	Programs the amplitude of the standard waveform in units of volts.
<offs>	-11 to 11	Numeric	0	Programs the offset of the standard waveform in units of volts.

Response

The 3172 returns the present frequency, amplitude and offset setting similar to the following example:

1e6,5,0.

APPLy:SINusoid <freq>,<ampl>,<offs>,<phas>,<power>(?)

Description

This command changes the waveform function to standard sine and programs the frequency, amplitude, offset, start phase and power coefficient simultaneously. This command affects the output regardless of the current output function. For example, if you generate FM, the 3172 will stop generating FM, will revert to the standard sine waveform and will update the sine parameters, as specified by this command.

Parameters

Name	Range	Type	Default	Description
<freq>	10e-3 to 30e6	Numeric	1e6	Programs the frequency of the standard sine waveform in units of Hz.
<ampl>	10e-3 to 22	Numeric	5	Programs the amplitude of the standard sine waveform in units of volts.
<offs>	-11 to 11	Numeric	0	Programs the offset of the standard sine waveform in units of volts.
<phas>	0 to 360	Numeric	0	Programs the phase start of the standard sine waveform in units of percent.
<power>	1 to 9	Numeric (integer only)	0	Programs the power coefficient of the standard sine waveform.

Response

The 3172 returns the present frequency, amplitude, offset, phase and power coefficient settings similar to the following example: 1e6,5,0,0,1.

APPLy:TRIangle <freq>,<ampl>,<offs>,<phas>,<power>(?)

Description

This command changes the waveform function to standard triangle and programs the frequency, amplitude, offset, start phase and power coefficient simultaneously. This command affects the output regardless of the current output function. For example, if you generate FM, the 3172 will stop generating FM, will revert to the standard triangle waveform and will update the triangle parameters, as specified by this command.

Parameters

Name	Range	Type	Default	Description
<freq>	10e-3 to 30e6	Numeric	1e6	Programs the frequency of the standard triangle waveform in units of Hz.
<ampl>	10e-3 to 22	Numeric	5	Programs the amplitude of the standard triangle waveform in units of volts.
<offs>	-11 to 11	Numeric	0	Programs the offset of the standard triangle waveform in units of volts.
<phas>	0 to 360	Numeric	0	Programs the phase start of the standard triangle waveform in units of percent.
<power>	1 to 9	Numeric (integer only)	0	Programs the power coefficient of the standard triangle waveform.

Response

The 3172 returns the present frequency, amplitude, offset, phase and power coefficient settings similar to the following example: 1e6,5,0,0,1.

APPLy:SQUare <freq>,<ampl>,<offs>,<dcycle>(?)

Description

This command changes the waveform function to standard square and programs the frequency, amplitude, offset, duty cycle simultaneously. This command affects the output regardless of the current output function. For example, if you generate FM, the 3172 will stop generating FM, will revert to the standard square waveform and will update the square parameters, as specified by this command.

Parameters

Name	Range	Type	Default	Description
<freq>	10e-3 to 30e6	Numeric	1e6	Programs the frequency of the standard square waveform in units of Hz.
<ampl>	10e-3 to 22	Numeric	5	Programs the amplitude of the standard square waveform in units of volts.
<offs>	-11 to 11	Numeric	0	Programs the offset of the standard square waveform in units of volts.
<dcycle>	0 to 99.99	Numeric	50	Programs the duty cycle of the standard square waveform in units of percent.

Response

The 3172 returns the present frequency, amplitude, offset, and duty cycle settings similar to the following example: 1e6,5,0,50.

APPLy:PULSe <freq>,<ampl>,<offs>,<del>,<wid>,<lee>,<tre>(?)

Description

This command changes the waveform function to standard pulse and programs the frequency, amplitude, offset, delay time, pulse width and leading and trailing edges simultaneously. This command affects the output regardless of the current output function. For example, if you generate arbitrary waveforms, the 3172 will stop generating arbitrary waveforms, will revert to the standard pulse waveform and will update the pulse parameters, as specified by this command.

Parameters

Name	Range	Type	Default	Description
<freq>	10e-3 to 30e6	Numeric	1e6	Programs the frequency of the standard pulse waveform in units of Hz.
<ampl>	10e-3 to 22	Numeric	5	Programs the amplitude of the standard pulse waveform in units of volts.
<offs>	-11 to 11	Numeric	0	Programs the offset of the standard pulse waveform in units of volts.
<del>	0 to 99.99	Numeric	10	Programs the delay time of the standard pulse waveform in units of percent.
<wid>	0 to 99.99	Numeric	10	Programs the pulse width of the standard pulse waveform in units of percent.
<lee>	0 to 99.99	Numeric	10	Programs the leading edge transition time of the standard pulse waveform in units of percent.
<tre>	0 to 99.99	Numeric	10	Programs the trailing edge transition time of the standard pulse waveform in units of percent.

Response

The 3172 returns the present frequency, amplitude, offset, delay time, pulse width and leading and trailing edges settings similar to the following example: 1e6,5,0,10,10,10,10.

APPLy:RAMP <freq>,<ampl>,<offs>,<del>,<lee>,<tre>(?)

Description

This command changes the waveform function to standard ramp and programs the frequency, amplitude, offset, delay time, pulse width and leading and trailing edges simultaneously. This command affects the output regardless of the current output function. For example, if you generate arbitrary waveforms, the 3172 will stop generating arbitrary waveforms, will revert to the standard ramp waveform and will update the pulse parameters, as specified by this command.

Parameters

Name	Range	Type	Default	Description
<freq>	10e-3 to 30e6	Numeric	1e6	Programs the frequency of the standard ramp waveform in units of Hz.
<ampl>	10e-3 to 22	Numeric	5	Programs the amplitude of the standard ramp waveform in units of volts.
<offs>	-11 to 11	Numeric	0	Programs the offset of the standard ramp waveform in units of volts.
<del>	0 to 99.99	Numeric	10	Programs the delay time of the standard ramp waveform in units of percent.
<lee>	0 to 99.99	Numeric	10	Programs the leading edge transition time of the standard ramp waveform in units of percent.
<tre>	0 to 99.99	Numeric	10	Programs the trailing edge transition time of the standard ramp waveform in units of percent.

Response

The 3172 returns the present frequency, amplitude, offset, delay time, pulse width and leading and trailing edges settings similar to the following example: 1e6,5,0,10,10,10.

APPLy:SINC <freq>,<ampl>,<offs>,<cycles>(?)

Description

This command changes the waveform function to standard sinc and programs the frequency, amplitude, offset and “0” crossing cycles simultaneously. This command affects the output regardless of the current output function. For example, if you generate modulated waveforms, the 3172 will stop generating these waveforms, will revert to the standard square waveform and will update the sinc parameters, as specified by this command.

Parameters

Name	Range	Type	Default	Description
<freq>	10e-3 to 30e6	Numeric	1e6	Programs the frequency of the standard sinc waveform in units of Hz.
<ampl>	10e-3 to 22	Numeric	5	Programs the amplitude of the standard sinc waveform in units of volts.
<offs>	-11 to 11	Numeric	0	Programs the offset of the standard sinc waveform in units of volts.
<cycles>	4 to 100	Integer	10	Programs the “0” crossing number of cycles of the standard sinc waveform in units of percent.

Response

The 3172 returns the present frequency, amplitude, offset and number of “0” crossing cycles settings similar to the following example: 1e6,5,0,10.

APPLy:GAUSsian <freq>,<ampl>,<offs>,<exp>(?)

Description

This command changes the waveform function to standard Gaussian and programs the frequency, amplitude, offset and exponent simultaneously. This command affects the output regardless of the current output function. For example, if you generate modulated waveforms, the 3172 will stop generating these waveforms, will revert to the standard Gaussian waveform and will update the Gaussian parameters, as specified by this command.

Parameters

Name	Range	Type	Default	Description
<freq>	10e-3 to 30e6	Numeric	1e6	Programs the frequency of the standard Gaussian waveform in units of Hz.
<ampl>	10e-3 to 22	Numeric	5	Programs the amplitude of the standard Gaussian waveform in units of volts.
<offs>	-11 to 11	Numeric	0	Programs the offset of the standard Gaussian waveform in units of volts.
<exp>	4 to 100	Integer	10	Programs the exponent of the standard Gaussian waveform.

Response

The 3172 returns the present frequency, amplitude, offset and exponent settings similar to the following example: 1e6,5,0,10.

APPLy:EXPonential <freq>,<ampl>,<offs>,<exp>(?)

Description

This command changes the waveform function to standard exponential and programs the frequency, amplitude, offset and exponent simultaneously. This command affects the output regardless of the current output function. For example, if you generate modulated waveforms, the 3172 will stop generating these waveforms, will revert to the standard exponential waveform and will update the exponential parameters, as specified by this command.

Parameters

Name	Range	Type	Default	Description
<freq>	10e-3 to 30e6	Numeric	1e6	Programs the frequency of the standard exponential waveform in units of Hz.
<ampl>	10e-3 to 22	Numeric	5	Programs the amplitude of the standard exponential waveform in units of volts.
<offs>	-11 to 11	Numeric	0	Programs the offset of the standard exponential waveform in units of volts.
<exp>	-100 to 100	Integer	100	Programs the exponent of the standard exponential waveform.

Response

The 3172 returns the present frequency, amplitude, offset and exponent settings similar to the following example: 1e6,5,0,100.

APPLy:DC <ampl>(?)

Description

This command changes the waveform function to standard dc and programs the amplitude simultaneously. This command affects the output regardless of the current output function. For example, if you generate modulated waveforms, the 3172 will stop generating these waveforms, will revert to the standard dc waveform and will update the dc parameters, as specified by this command.

Parameters

Name	Range	Type	Default	Description
<dc_ampl>	-100 to 100	Numeric	100	Programs the amplitude of the standard dc waveform in units of percent.

Response

The 3172 returns the present frequency, amplitude, offset and exponent settings similar to the following example: 100.

APPLy:USER <seg_#><sclk>,<ampl>,<offs>(?)

Description

This command changes the waveform function to arbitrary and programs the active segment sample clock, amplitude and offset simultaneously. This command affects the output regardless of the current output function. For example, if you generate standard waveforms, the 3172 will stop generating these waveforms, will revert to the arbitrary function and will update the arbitrary waveform parameters, as specified by this command.

Parameters

Name	Range	Type	Default	Description
<seg_#>	1 to 10k	Integer	1	Selects the active segment number of the arbitrary waveform.
<sclk>	10e-6 to 200e6	Numeric	1e6	Programs the sample clock frequency of the arbitrary waveform in units of S/s.
<ampl>	10e-3 to 22	Numeric	5	Programs the amplitude of the arbitrary waveform in units of volts.
<offs>	-11 to 11	Numeric	0	Programs the offset of the arbitrary waveform in units of volts.

Response

The 3172 returns the present segment number, sample clock, amplitude and offset settings similar to the following example: 1,1e6,5,0.

Sequenced Waveform Control Commands (W6 Modules Only)

This group is used to control the sequenced waveforms and their respective parameters. This will allow you to create multiple sequence table and modify segment loops and links. Also use these commands to add or delete sequences from your instrument.

Table 5-8 summarizes the sequenced waveforms control commands. Factory defaults after *RST are shown in the Default column. Parameter range and low and high limits are listed, where applicable.

Generating Sequenced Waveforms

Sequenced waveforms are made of a number of arbitrary waveforms, which can be linked and looped in user-programmable order. Sequenced waveforms are generated from waveforms stored in the 3172 as memory segments. Therefore, before a sequence can be used, download waveform segments to the arbitrary memory using TRAC# or DMA methods. Information on how to partition the memory and how to download waveforms is given in Chapter 3 in the section titled **Generating Arbitrary Waveforms**.

An example of how sequenced waveforms work is demonstrated in Figures 1-7 through 1-9. The sequence generator lets you link and loop segments in user-defined order. Figure 1-10 shows a sequence of waveforms that were stored in three different memory segments.

There are a number of tools that you can use to build a sequence table. The easiest way is of course to use the ArbConnection program. Information how to use the ArbConnection program is given in a later chapter. In other cases, SCPI programming allows low-level programming of sequence tables.

In general, sequences can be build one step at a time using the SEQ:DEF command. The one step method is slow and tedious however, it allows better control for one who just begins his first sequence programming. Advanced users can download a complete sequence table using the binary sequence download option. The latter being much faster for applications requiring large sequence tables. Use the information below to understand sequence commands and how to implement them in your application.

Table 8, Sequence Control Commands

Keyword	Parameter Form	Default	Association
[:SOURce]			
:SEQuence			
[:DATA]	<data_array>		W6
:ADVance	AUTOmatic | TRIGgered | STEP | MIX	AUTO	W6
:SELect	1 to 10	1	W6
:DEFine	<step>,<seg_#>,<repeat>,<adv_mode_x>,<sync_bit_x>		W6
:DELete			
[:NAME]	1 to 4096		W6
:ALL			W6
:SYNC			
[:TYPe]	LCOMplete | BIT	LCOM	W6

SEQuence #<header><binary_block>

Description

This command will build a complete sequence table in one binary download. In this way, there is no need to define and download individual sequencer steps. Using this command, sequence table data is loaded to the 3172 using high-speed binary transfer in a similar way to downloading waveform data with the trace command. High-speed binary transfer allows any 8-bit bytes (including extended ASCII code) to be transmitted in a message. This command is particularly useful for long sequences that use a large number of segment and sequence steps. As an example, the next command will generate three-step sequence with 16 bytes of data that contains segment number, repeats (loops) and mixed mode flag option.

```
SEQuence#216<binary_block>
```

This command causes the transfer of 16 bytes of data (2-step sequence) to the sequence table buffer. The <header> is interpreted this way:

- The ASCII "#" ($23) designates the start of the binary data block.
- "2" designates the number of digits that follow.
- "16" is the number of bytes to follow. This number must divide by 8.

The generator accepts binary data as 64-bit integers, which are sent in two-byte words. Therefore, the total number of bytes is always eight times the number of sequence steps. For example, 16 bytes are required to download 2 sequence steps to the sequence table. The IEEE-STD-488.2 definition of Definite Length Arbitrary Block Data format is demonstrated in Figure 5-2. The transfer of definite length arbitrary block data must terminate with the EOI bit set. This way, carriage-return (CR – 0dH) and line feed (LF – 0aH) characters can be used as sequence data and will not cause unexpected termination of the arbitrary block data. Figure 5-5 shows how to prepare the 64-bit word for the sequence step, repeat, mixed mode and sync bit.

The sequence table data is made of 64-bit words, however, the data has to be prepared as 64-bit words and rearranged as six 8-bit words before it can be used by the 3172 as sequence table data. Figure 5-5 shows how to prepare the 64-bit word for the sequence step, repeat, and mixed mode flag.

There are a number of points you should be aware of before you start preparing the data:

1. Each channel has its own sequence table buffer. Therefore, make sure you selected the correct active channel (with the INST:SEL command) before you download sequence table data to the generator
2. Minimum number of sequencer steps is 1; maximum number is 4096
3. The number of bytes in a complete sequence table must divide by 8. The 3172 has no control over data sent to its sequence table during data transfer. Therefore, wrong data and/or incorrect number of bytes will cause erroneous sequence partition
4. The LSB bit is the only bit used in the mode byte. This bit has an effect on the operation of the sequence only when Mixed Step Advance mode is active. With the LSB bit set to “0”, the sequence generator will advance to the next step automatically. With the LSB bit set to “1”, the sequence generator will advance to the next step only when a valid trigger signal is sensed at the trigger input.
5. SYNC state bit is valid only when the sequence sync type is BIT

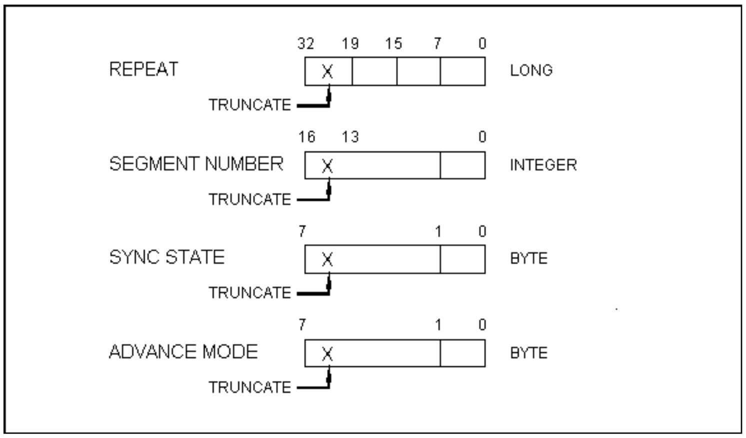

Figure 5-5, 64-bit Sequence Table Download Format

Parameters

Name	**Type**	**Description**
<binary_block>	Binary	Block of binary data that contains information on the sequence table.

SEQuence:ADVance {AUTOmatic|STEP|SINGle|MIXed}(?)

Description

This command selects the sequence advance mode. The way the instrument advances through the sequence links can be specified by the user.

Parameters

Name	**Type**	**Default**	**Description**
AUTOmatic	Discrete	AUTO	Specifies continuous advance where the generator steps continuously to the end of the sequence table and repeats the sequence from the start. For example, if a sequence is made of three segments 1, 2 and 3, the sequence will generate an infinite number of 1,2,3,1,2,3,1,2,3…waveforms. Of course, each link (segment) can be programmed with its associated loop (repeat) number.

TRIGgered	Discrete	In triggered advance mode, the generator idles between steps until a valid trigger signal is sensed. This mode operates with trigger mode only. An attempt to select the TRIG advance mode when the 3172 is in continuous operating mode will generate an error. After trigger, the generator outputs one waveform cycle. Then, the output level idles at a DC level equal to the last point of the last generated waveform. If loops (repeats) were programmed, the output will repeat this segment every time a trigger is received. Only after executing all of the programmed loops will the sequence step to the next assigned segment.
STEP	Discrete	In step advance mode, the sequence is advanced to the next waveform only when a valid trigger is received. The output of the 3172 generates the first segment continuously until a trigger signal advances the sequence to the next segment. If repeats were selected for a segment, the loop counter is executed automatically.
MIXed	Discrete	Mixed mode is a special mode that combines continuous step advance with single step advance in a sequence. There are three conditions for the sequence generator to operate in this mode: 1) The 3172 is set to operate in continuous mode 2) Select the MIX sequence advance mode 3) Assign the mixed mode bits for each sequence step in your SEQ:DEF command. “0” programs normal advance, “1” programs trigger advance. Step with a “0” bit assigned to it will advance automatically to the next step. If “1” is assigned to a step, the instrument will generate this step and its associated number of repeats continuously and only a valid trigger signal will advance this step to the next step.

Response

The 3172 returns the AUTO, TRIG, STEP, or MIX depending on the present sequence advance mode setting.

SEQuence:SELect <sequence_#>(?)

Description

This command selects an active sequence to be generated at the output connector. By selecting the active sequence, successive :SEQ commands will affect the selected sequence only

Parameters

Name	Range	Type	Default	Description
<sequence_# >	1 to 10	Numeric (integer only)	1	Selects the active sequence number

Response

The 3172 returns the active sequence number.

SEQuence:DEFine <step>,<seg_#>,<repeat>,<adv_mode>,<sync_bit>(?)

Description

This command builds a step in a sequence table. It defines all of the parameters that are associated with the sequence step such as segment number, link, loop, advance mode and sync mode.

Parameters

Name	Range	Type	Description
<step>	1 to 4096	Numeric (integer only)	Programs the step in the sequence table. Steps are indexed from 1 to 4096 and must be programmed in an ascending order; Empty step locations in a sequence table are not permitted.
<seg_#>	1 to 10k	Numeric (integer only)	Assigns a segment to a specific step number. When encountered in the sequence table, the segment number that is associated with the step is generated.
<repeat>	1 to 1M	Numeric integer only)	Programs the repeat number of loops that a specific step will play before advancing to the next step in the sequence.
<adv_mode>	0-1	Boolean	A “0” programs normal advance, a “1” programs trigger advance. Steps with a “0” assigned to it will advance automatically to the next step. If “1” is assigned to a step, the instrument will generate this step continuously and only a valid trigger signal will cause the sequence to advance to the next step. Note that the <adv_mode> parameter has no effect when the sequence advance mode is set to SING
<sync_bit>	0-1	Boolean	A “1” programs a sync bit to be present at a specific sequence step. This feature is useful for applications requiring multiple sync bits in a single sequence. Note that normal sync output during sequence mode is LCOM.

NOTE

Although trigger signals are used to advance mixed mode, the mixed mode operates in continuous mode only. The <mode> parameter is ignored if you use SING as the advance mode for the sequence table.

TIP

Every time you use the SEQ:DEF command while your 3172 is in sequenced operating mode, the instrument attempts to rebuild the sequence table and restart the sequence. Therefore, sending this command in sequenced mode will slow the programming process and the operation of the generator. Using the SEQ:DEF command in FIX or USER mode will greatly speed up programming time.

SEQuence:DELete <sequence_# >

Description

This command will delete a step in a specific sequence table. Before you use this step make sure your sequence number is setup correctly for this operation.

Parameters

Name	Range	Type	Default	Description
<sequence_# >	1 to 4096	Numeric (integer only)	1	Selects the step number which will be deleted

SEQuence:DELete:ALL

Description

This command will delete the entire sequence table. Before you use this step make sure your sequence number is setup correctly for this operation.

OUTPut:SYNC:TYPE {LCOMplete|BIT }(?)

Description

This command programs the 3172 SYNC mode.

Parameters

Name	Type	Default	Description
LCOMplete	Discrete	LCOM	The sync output will transition high at the beginning of the sequence and will transition low at the end of the sequence, less 16 waveform points.
BIT	Discrete		The sync output will generate a pulse at the beginning of a specific segment regardless how many times the segment appears in a sequence. The width of the sync pulse is 16 waveform points.

Response

The 3172 returns LCOM or BIT depending on the present SYNC type.

Modulated Waveform Global Control Commands (W6 Module Only)

This group is used to set up the instrument in modulated waveforms mode and to select the general parameters that control all of modulation functions. Note that the modulation can be turned off to create continuous carrier waveform (CW). The following modulation schemes can be selected and controlled: AM, FM, Sweep, FSK, ASK, PSK, Amplitude and Frequency hops and 3D. Table 5-9 summarizes the modulated waveforms global commands. Factory defaults after *RST are shown in the Default column. Parameter range and low and high limits are listed, where applicable.

Table 9, Modulated Waveforms Global Commands

Keyword	Parameter Form	Default	Association
[:SOURce]			
:MODulation			
:TYPE	OFF | AM | FM | SWE | FSK | ASK | PSK | FHOPping | AHOPping | EXAM	OFF	W6
:CARRier			
[:FREQuency]	10 to 30e6	1e6	W6
:BASeline	CARRier | DC	CARR	W6

MODulation:TYPE {OFF|AM|FM|SWEeep|FSK|ASK|PSK|FHOPping|AHOPping}(?)

Description

This command selects the modulation type. All modulation types are internal, thus external signals are not required for producing modulation.

Parameters

Name	Type	Default	Description
OFF	Discrete	OFF	Modulation off is a special mode where the output generates continuous, non-modulated sinusoidal carrier waveform (CW).
AM	Discrete		This turns on the AM function. Program the AM parameters to fine tune the function for your application.
FM	Discrete		This turns on the FM function. Program the FM parameters to fine tune the function for your application.
SWEep	Discrete		This turns on the sweep function. Program the sweep parameters to fine tune the function for your application.
FSK	Discrete		This turns on the FSK function. Program the FSK parameters to fine tune the function for your application.
ASK	Discrete		This turns on the ASK function. Program the ASK parameters to fine tune the function for your application.
PSK	Discrete		This turns on the PSK function. Program the PSK parameters to fine tune the function for your application.

FHOPping	Discrete	This turns on the frequency hopping function. Program the hop parameters to fine tune the function for your application.
AHOPping	Discrete	This turns on the amplitude hopping function. Program the amplitude hopping parameters to fine tune the function for your application.

Response

The 3172 returns OFF, AM, FM, SWE, FSK, ASK, PSK, FHOP, or AHOP depending on the present modulation type setting.

MODulation:CARRier <frequency>(?)

Description

This command programs the CW frequency. Note that the CW waveform is sine only and its frequency setting is separate to the standard sine waveform. The CW frequency setting is valid for all modulation types.

Parameters

Name	Range	Type	Default	Description
<frequency>	10e-3 to 30e6	Numeric	1e6	Programs the frequency of the carrier waveform in units of Hz. Note that the CW waveform is sine only and its frequency setting is separate to the standard sine waveform.

Response

The 3172 returns the current carrier frequency value.

MODulation:CARRier:BASeline {CARRier|DC}(?)

Description

This command programs the carrier baseline when the modulation is used in triggered mode.

Parameters

Name	Type	Default	Description
CARRier	Discrete	CARR	This selects the carrier as the baseline for the modulation function, when operating in one of the interrupted run modes. The output will generate continuous, none modulated sinusoidal waveform (CW) until triggered, upon trigger will generate the modulated waveform and then resume generating continuous CW.
DC	Discrete		This selects DC level as the baseline for the modulation function, when operating in one of the interrupted run modes. The output will generate continuous DC until triggered, upon trigger will generate the modulated waveform and then resume generating continuous DC level.

Response

The 3172 returns CARR, or DC depending on the present carrier baseline setting.

Modulation Control Commands (W6 Module Only)

This group is used to control parameters for individual modulation schemes. Control parameters are available for: AM, FM, Sweep, FSK, ASK, PSK, Amplitude and Frequency hops and 3D. The modulation control commands are summarized in Table 5-10. Factory defaults after *RST are shown in the Default column. Parameter range and low and high limits are listed, where applicable.

Table 10, Modulated Waveform Control Commands

Keyword	Parameter Form	Default	Association
[:SOURce]			
:AM			
:FUNCtion			
:SHAPe	SINusoid | TRIangle | SQUare | RAMP	SIN	W6
:INTernal			
:FREQuency	10e-3 to 1e6	100	W6
:DEPTh	0 to 100	50	W6
:EXEcute		(= MOD:TYPE AM)	W6
:FM			
:DEViation	10.0e-3 to 30e6	100e3	W6
:FUNCtion			
:SHAPe	SINusoid | TRIangle | SQUare | RAMP | ARB	SIN	W6
:FREQuency	10e-3 to 350e3	10e3	W6
:RASTer	1 to 2.5e6	1e6	W6
:MARKer			
[:FREQuency]	10e-3 to 30e6	1e6	W6
:DATA	<data_array>		W6
:SWEep			
:FREQuency			
[:STARt]	10 to 30e6	10e3	W6
:STOP	10 to 30e6	1e6	W6
:RASTer	10e-6 to 200e6 | MINimum | MAXimum	1e6	W6
:FUNCtion	SINusoidal | TRIangle(*) | SQUare(*)	SIN (*) Computed	W6
:TIME	1.4e-6 to 40.0	1e-3	W6
:DIRection	UP | DOWN	UP	W6
:SPACing	LINear | LOGarithmic	LIN	W6
:MARKer			
[:FREQuency]	10 to 30e6	505e3	W6
:FSK			
:FREQuency			

Table 5-10, Modulated Waveform Control Commands (continued)

Keyword	Parameter Form	Default	Association
:SHIFted	10e-3 to 30e6	100e3	W6
:BAUD	1 to 10e6	10e3	W6
:MARKer	1 to 4000	1	W6
:DATA	<data_array>		W6
[:SOURce]			
:ASK			
[:AMPLitude]			
[:STARt]	0 to 16	5	W6
:SHIFted	0 to 16	1	W6
:BAUD	1 to 10e6	10e3	W6
:MARKer	1 to 1000	1	W6
:DATA	<data_array>		W6
:PSK			
:PHASe			
[:STARt]	0 to 360	0	W6
:SHIFted	0 to 360	180	W6
:RATE	1 to 10e6	10e3	W6
:DATA	<data_array>		W6
:MARKer	1 to 4000	1	W6
:FHOPping			
:DWELl			
:MODe	FIXed | VARiable	FIX	W6
[:TIMe]	200e-9 to 20	200e-9	W6
:FIXed			
:DATA	<data_array>		W6
:VARiable			
:DATA	<data_array>		W6
:MARKer	1 to 5000	1	W6
:AHOPping			
:DWELl			
:MODe	FIXed | VARiable	FIX	W6
[:TIMe]	200e-9 to 20	200e-9	W6
:FIXed			
:DATA	<data_array>		W6
:VARiable			
:DATA	<data_array>		W6
:MARKer	1 to 5000	1	W6

AM Programming

Use the following commands for programming the internal AM parameters. AM control is internal for this function; the external AM function is described in a different section. The commands for programming the internal amplitude modulation function are described below. Note that the carrier waveform frequency (CW) setting is common to all modulation schemes.

AM:FUNCtion:SHAPe {SINusoid|TRIangle|SQUare|RAMP}(?)

Description

This command selects one of the waveform shapes as the active modulating waveform.

Parameters

Name	Type	Default	Description
SINusoid	Discrete	SIN	Selects the sine shape as the modulating waveform
TRIangle	Discrete		Select the triangular shape as the modulating waveform
SQUare	Discrete		Select the square shape as the modulating waveform
RAMP	Discrete		Selects the ramp shape as the modulating waveform

Response

The 3172 returns SIN, TRI, SQU, or RAMP depending on the selected function shape setting.

AM:INTernal:FREQuency <am_freq>(?)

Description

This command sets the modulating wave frequency for the built-in standard modulating waveform library.

Parameters

Name	Range	Type	Default	Description
<am_freq>	10e-3 to 1e6	Numeric	100	Programs the frequency of the modulating waveform in units of Hz. The frequency of the built-in standard modulating waveforms only is affected.

Response

The 3172 returns the present modulating waveform frequency value. The returned value is in standard scientific format (for example: 100mHz would be returned as 100e-3 – positive numbers are unsigned).

AM:DEPth <depth>(?)

Description

This command sets the modulating wave frequency for the built-in standard modulating waveform library.

Parameters

Name	Range	Type	Default	Description
<depth>	0 to 100	Numeric	50	Programs the depth of the modulating waveform in units of percent.

Response

The 3172 returns the present modulating depth value.

FM Modulation Programming

Use the following commands for programming the FM parameters. FM control is internal. There are two types of waveforms that can be used as the modulating waveforms: Standard and Arbitrary. The standard waveforms are built in a library of waveforms and could be used anytime without external control. The arbitrary waveforms must be loaded into a special FM arbitrary waveform memory and only then can be used as a modulating waveform.

FM:DEViation <deviation>(?)

Description

This programs the deviation range around the carrier frequency. The deviation range is always symmetrical about the carrier frequency. If you need non-symmetrical deviation range, you can use the arbitrary FM composer screen or an external utility to design such waveforms.

Parameters

Name	Range	Type	Default	Description
<deviation>	10e-3 to 30e6	Numeric	100e3	Programs the deviation range around the carrier frequency in units of Hz.

Response

The 3172 returns the present deviation frequency value. The returned value will be in standard scientific format (for example: 100mHz would be returned as 100e-3 – positive numbers are unsigned).

FM:FUNCtion:SHAPe {SINusoid|TRIangle|SQUare|RAMP|ARB}(?)

Description

This command selects one of the waveform shapes as the active modulating waveform.

Parameters

Name	Type	Default	Description
SINusoid	Discrete	SIN	Selects the sine shape as the modulating waveform
TRIangle	Discrete		Select the triangular shape as the modulating waveform

SQUare	Discrete	Select the square shape as the modulating waveform
RAMP	Discrete	Selects the ramp shape as the modulating waveform
ARB	Discrete	Selects an arbitrary waveform as the modulating shape. The waveform must be designed and downloaded to the FM arbitrary modulating waveform memory before one can use this option. Information on how to create and download FM arbitrary waveforms is given later in this chapter.

Response

The 3172 returns SIN, TRI, SQU, RAMP, or ARB depending on the selected function shape setting.

FM:FREQuency <fm_freq>(?)

Description

This command sets the modulating wave frequency for the built-in standard modulating waveform library.

Parameters

Name	Range	Type	Default	Description
<fm_freq>	10e-3 to 350e3	Numeric	10e3	Programs the frequency of the modulating waveform in units of Hz. The frequency of the built-in standard modulating waveforms only is affected.

Response

The 3172 returns the present modulating waveform frequency value. The returned value will be in standard scientific format (for example: 100mHz would be returned as 100e-3 – positive numbers are unsigned).

FM:FREQuency:RASTer <arb_fm_freq>(?)

Description

This command sets the sample clock frequency for the arbitrary modulating waveform. Arbitrary modulating waveforms must be created in an external utility and downloaded to the FM arbitrary waveform memory before this function can be used.

Parameters

Name	Range	Type	Default	Description
<arb_fm_freq>	1 to 2.5e6	Numeric	1e6	Programs the sample clock frequency of the arbitrary modulating waveform in units of S/s.

Response

The 3172 returns the present sample clock of the arbitrary modulating waveform value. The returned value will be in standard scientific format (for example: 100mHz would be returned as 100e-3 – positive numbers are unsigned).

FM:MARKer <frequency>(?)

Description

This function programs marker frequency position. FM marker can be placed inside the following range: (carrier frequency ± deviation frequency / 2). The marker pulse is output from the SYNC output connector.

Parameters

Name	Range	Type	Default	Description
<frequency>	10e-3 to 30e6	Numeric	1e6	Programs the marker frequency position in units of Hz.

Response

The 3172 returns the present marker frequency value. The returned value will be in standard scientific format (for example: 100mHz would be returned as 100e-3 – positive numbers are unsigned).

FM:DATA #<header><binary_block>

Description

This command downloads FM modulating waveform data to the arbitrary FM memory. Arbitrary modulating waveform table data is loaded to the 3172 using high-speed binary transfer in a similar way to downloading waveform data with the trace command. High-speed binary transfer allows any 8-bit bytes (including extended ASCII code) to be transmitted in a message. Remember, downloading data to the arbitrary FM waveform memory is very different than loading arbitrary waveform data. Waveform data programs amplitude domain therefore, every point programs an amplitude level. On the other hand, FM modulating waveform data programs frequency domain therefore, every point sets different sample clock frequency.

```
FM:DATA#3100<binary_block>
```

This command causes the transfer of 10 bytes of data to the arbitrary FM waveform memory. The <header> is interpreted this way:

- The ASCII "#" ($23) designates the start of the binary data block.
- "3" designates the number of digits that follow.
- "100" is the number of bytes to follow. This number must divide by 4.

The generator accepts binary data as 32-bit integers, which are sent in five-byte words. Therefore, the total number of bytes is always three times the number of arbitrary FM waveform points. For example, 100 bytes are required to download 20 arbitrary FM waveform points. The IEEE-STD-488.2 definition of Definite Length Arbitrary Block Data format is demonstrated in Figure 5-2 (refer to the TRACe subsystem). The transfer of definite length arbitrary block data must terminate with the EOI bit set. This way, carriage-return (CR – 0dH) and line feed (LF – 0aH) characters can be used as sequence data and will not cause unexpected termination of the arbitrary block data.

Downloading data to the arbitrary FM waveform memory is very different than loading arbitrary waveform data. Waveform data programs in the amplitude domain such that every point programs an amplitude level. On the other hand, FM modulating waveform data programs in the frequency domain such that every point sets a different frequency. The FM modulating waveform data is made of 32-bit words. The data has to be prepared as 32-bit words and rearranged as five 8-bit words before it can be used by the 3172 as FM modulating waveform data.

There are a number of points you should be aware of before you start preparing the data:

1. The number of bytes in a complete FM modulating waveform data must divide by 4. The 3172 has no control over data sent to its FM waveform during data transfer. Therefore, wrong data and/or incorrect number of bytes will cause errors
2. The LSB on the last byte sets marker position. “0” = sets no marker and “1” sets marker. You can set as many markers as you want.
3. The SYNC output serves as marker output when you have the 3172 set to operate in FM mode. Normal SYNC level is TTL low. The SYNC output is set to TTL high at the position of the marker. This way you can use the SYNC output to mark frequency occurrences during FM operation.
4. Data download is terminated with the MSB of the last byte set to 1.

The following sequence should be used for downloading arbitrary FM Waveforms:

1. Prepare your FM waveform data points using the following relationship:

   N = Frequency[Hz] x 14.31655765

2. Use an I/O routine such as ViMoveAsync (from the VISA I/O library) to transfer binary blocks of data to the generator.

Parameters

Name	**Type**	**Description**
<binary_block>	Binary	Block of binary data that contains information on the arbitrary modulating waveform.

Sweep Modulation Programming

Use the following commands for programming the sweep parameters. Sweep control is internal. The frequency will sweep from start to stop frequencies at an interval determined by the sweep time value and controlled by a step type determined by the sweep step parameter.

There are two sweep modes: Linear, where the step of which the generator increments from start to stop frequency is linear and Logarithmic, where the step of which the generator increments from start to stop frequency is logarithmic

The commands for programming the frequency sweep function are described below.

SWEep:FREQuency <start_freq>(?)

Description

This specifies the sweep start frequency. The 3172 normally sweeps from start to stop frequencies, however, if the sweep direction is reversed, the output will sweep from stop to start frequencies. The start and stop frequencies may be programmed freely throughout the frequency of the standard waveform frequency range.

Parameters

Name	**Range**	**Type**	**Default**	**Description**
<start_freq>	10e-3 to 30e6	Numeric	10e3	Programs the sweep start frequency. Sweep start is programmed in units of Hz.

Response

The 3172 returns the present sweep start frequency value. The returned value will be in standard scientific format (for example: 100mHz would be returned as 100e-3 – positive numbers are unsigned).

SWEep:FREQuency:STOP <stop_freq>(?)

Description

This specifies the sweep stop frequency. The 3172 will normally sweep from start to stop frequencies however, if the sweep direction is reversed, the output will sweep from stop to start frequencies. The start and stop frequencies may be programmed freely throughout the frequency of the standard waveform frequency range.

Parameters

Name	Range	Type	Default	Description
<stop_freq>	10e-3 to 30e6	Numeric	1e6	Programs the sweep stop frequency. Sweep stop is programmed in units of Hz.

Response

The 3172 returns the present sweep stop frequency value. The returned value will be in standard scientific format (for example: 100mHz would be returned as 100e-3 – positive numbers are unsigned).

SWEep:FREQuency:RASTer <sclk_freq>(?)

Description

This programs the sample clock frequency for the swept waveform. Program this parameter only if you fully understand the effect on the waveform otherwise let the instrument determine the sample clock setting as required to successfully complete the setting of the sweep.

Parameters

Name	Range	Type	Default	Description
<sclk_freq>	10e-3 to 200e6	Numeric	1e6	Programs the sample clock frequency in units of samples per second.

Response

The 3172 returns the present sweep sample clock frequency value. The returned value will be in standard scientific format (for example: 100mHz would be returned as 100e-3 – positive numbers are unsigned).

SWEep:FUNCtion {SINusoid|TRIangle|SQUare}(?)

Description

This specifies the swept function. There are three functions that could be swept are: Sine, Triangle and Square. The sine sweep is generated by the DDS but the triangle and the square are computed and place in the memory as complete waveforms and replayed by the arbitrary mode as a regular arbitrary waveform.

Parameters

Name	Type	Default	Description
SINusoid	Discrete	SIN	Selects sine as the swept waveform
TRIangle	Discrete		Selects triangle as the swept waveform
SQUare	Discrete		Selects square as the swept waveform

Response

The 3172 returns SIN, TRI, or SQU depending on the selected waveform setting.

SWEep:TIMe <time>(?)

Description

This specifies the time that will take the 3172 to sweep from start to stop frequencies. The time does not depend on the sweep boundaries as it is automatically adjusted by the software to the required interval. At the end of the sweep cycle the output waveform maintains the sweep stop frequency setting except if the 3172 is in continuous run mode where the sweep repeats itself continuously.

Parameters

Name	Range	Type	Default	Description
<time>	1.4e-6 to 40	Numeric	1e-3	Programs the sweep time. Sweep time is programmed in units of s.

Response

The 3172 returns the present sweep time. The returned value will be in standard scientific format (for example: 100ms would be returned as 100e-3 – positive numbers are unsigned).

SWEep:DIRection {UP|DOWN}(?)

Description

This specifies if the 3172 output will sweep from start-to-stop (UP) or from stop-to-start (DOWN) frequencies. Sweep time does not affect the sweep direction and frequency limits. At the end of the sweep cycle the output waveform normally maintains the sweep stop frequency setting but will maintain the start frequency, if the DOWN option is selected except if the 3172 is in continuous run mode where the sweep repeats itself continuously.

Parameters

Name	Type	Default	Description
UP	Discrete	UP	Selects the sweep up direction
DOWN	Discrete		Select the sweep down direction

Response

The 3172 returns UP, or DOWN depending on the selected direction setting.

SWEep:SPACing {LINear|LOGarithmic}(?)

Description

This specifies the sweep step type. Two options are available: logarithmic or linear. In linear, the incremental steps between the frequencies are uniform throughout the sweep range. Logarithmic type defines logarithmic spacing throughout the sweep start and stop settings.

Parameters

Name	Type	Default	Description
LINear	Discrete	LIN	Selects the linear sweep spacing
LOGarithmic	Discrete		Select the logarithmic sweep spacing

Response

The 3172 returns LIN, or LOG depending on the selected spacing setting.

SWEep:STEP <#_steps>(?)

Description

This programs the number of steps for the swept waveform. Program this parameter only if you fully understand the effect on the waveform otherwise let the instrument determine the number of steps as required to successfully complete the setting of the sweep.

Parameters

Name	Range	Type	Default	Description
<#_steps>	10 to 2000	Numeric (integer only)	1e6	Programs the number of steps in a sweep. This number affects the swept triangle and square only.

Response

The 3172 returns the present number of sweep steps value.

SWEep:MARKer <frequency>(?)

Description

This function programs marker frequency position. Sweep marker can be placed in between the start and the stop frequencies. The marker pulse is output from the SYNC output connector.

Parameters

Name	Range	Type	Default	Description
<frequency>	10 to 30e6	Numeric	505e3	Programs the marker frequency position in units of Hz.

Response

The 3172 returns the present marker frequency value. The returned value will be in standard scientific format (for example: 100mHz would be returned as 100e-3 – positive numbers are unsigned).

FSK Modulation Programming

Use the following commands for programming the FSK parameters. FSK control is internal. The frequency will shift from carrier to shifted frequency setting at a rate determined by the baud value and controlled by a sequence of bits in the FSK data table. The commands for programming the frequency shift keying function are described below. Note that the carrier waveform frequency (CW) setting is common to all modulation schemes.

FSK:FREQuency:SHIFted <shift_freq>(?)

Description

This programs the shifted frequency. The frequency shifts when the pointer in the data array points to “1”.

Parameters

Name	Range	Type	Default	Description
<shift_freq>	10e-3 to 30e6	Numeric	100e3	Programs the shifted frequency value in units of Hz.

Response

The 3172 returns the present shifted frequency value. The returned value will be in standard scientific format (for example: 100mHz would be returned as 100e-3 – positive numbers are unsigned).

FSK:FREQuency:BAUD <baud>(?)

Description

This allows the user to select FSK word rate. The word rate is the interval of which the bit streams in the FSK data array are clocked causing the output frequency to hop from carrier to shifted frequency values and vice-versa.

Parameters

Name	Range	Type	Default	Description
<baud>	1 to 10e6	Numeric	10e3	Programs the rate of which the frequency shifts from carrier to shifted frequency in units of Hz.

Response

The 3172 returns the present baud value. The returned value will be in standard scientific format (for example: 100mHz would be returned as 100e-3 – positive numbers are unsigned).

FSK:FREQuency:MARKer <index>(?)

Description

Programs where on the data stream the 3172 will generate a pulse, designated as FSK marker, or index point. The marker pulse is generated at the SYNC output connector. Note that if you intend to program marker position, you must do it before you load the FSK data list.

Parameters

Name	Range	Type	Default	Description
<index>	1 to 4000	Numeric (integer only)	1	Programs a marker pulse at an index bit position.

Response

The 3172 returns the present marker position.

FSK:DATA <fsk_data>

Description

Loads the data stream that will cause the 3172 to hop from carrier to shifted frequency and vice-versa. Data format is a string of "0" and "1" values which define when the output is the carrier frequency and when it shifts frequency to the FSK value. "0" defines carrier frequency,"1" defines shifted frequency. Note that if you intend to program marker position, you must do it before you load the FSK data list.

Below you can see how an FSK data table is constructed. The sample below shows a list of 10 shifts. The 3172 will step through this list, outputting either carrier or shifted frequencies, depending on the data list: Zero will generate carrier frequency and One will generate shifted frequency. Note that the waveform is always sinewave and that the last cycle is always completed.

Sample FSK Data Array

0 1 1 1 0 1 0 0 0 1

Parameters

Name	Type	Description
<fsk_data>	ASCII	Block of ASCII data that contains information for the generator when to shift from carrier to shifted frequency and vice-versa.

ASK Modulation Programming

Use the following commands for programming the ASK parameters. ASK control is internal. The amplitude will toggle between two amplitude settings at a rate determined by the baud value and controlled by a sequence of bits in the ASK data table. The commands for programming the amplitude shift keying function are described below. Note that the carrier waveform frequency (CW) setting is common to all modulation schemes.

ASK <amplitude>(?)

Description

This programs the normal amplitude setting. The amplitude shifts when the pointer in the data array points to “1”.

Parameters

Name	Range	Type	Default	Description
<amplitude>	0 to 20	Numeric	5	Programs the amplitude setting in units of volt.

Response

The 3172 returns the present amplitude value. The returned value will be in standard scientific format (for example: 100mV would be returned as 100e-3 – positive numbers are unsigned).

ASK:SHIFted <shift_ampl>(?)

Description

This programs the shifted amplitude. The amplitude shifts when the pointer in the data array points to “1”.

Parameters

Name	Range	Type	Default	Description
<shift_ampl>	0 to 20	Numeric	1	Programs the shifted amplitude setting in units of volt.

Response

The 3172 returns the present shifted amplitude value. The returned value will be in standard scientific format (for example: 100mHz would be returned as 100e-3 – positive numbers are unsigned).

ASK:BAUD <rate>(?)

Description

This allows the user to select ASK word rate. The word rate is the interval at which the bit streams in the ASK data array are clocked causing the output amplitude to hop from one level to shifted amplitude level values and vice-versa.

Parameters

Name	Range	Type	Default	Description
<rate>	1 to 10e6	Numeric	10e3	Programs the rate of which the frequency shifts from carrier to shifted frequency in units of Hz.

Response

The 3172 returns the present baud value. The returned value will be in standard scientific format (for example: 100kHz would be returned as 100e3 – positive numbers are unsigned).

ASK:FREQuency:MARKer <index>(?)

Description

Programs where on the data stream the 3172 will generate a pulse designated as the ASK marker or index point. The marker pulse is generated at the SYNC output connector. Note that if you intend to program marker position, you must do it before you load the ASK data list.

Parameters

Name	Range	Type	Default	Description
<index>	1 to 1000	Numeric (integer only)	1	Programs a marker pulse at an index bit position.

Response

The 3172 returns the present marker position.

ASK:DATA <ask_data>

Description

Loads the data stream that will cause the 3172 to hop from one amplitude level to shifted amplitude level and vice-versa. Data format is a string of "0" and "1" which define when the output generates base level and when it shifts amplitude to the ASK value. "0" defines base level amplitude,"1" defines shifted amplitude level. Note that if you intend to program marker position, you must do it before you load the ASK data list.

Below you can see how an ASK data table is constructed. The sample below shows a list of 10 shifts. The 3172 will step through this list, outputting either base or shifted amplitudes, depending on the data list: Zero will generate base level and One will generate shifted amplitude. Note that the waveform is always sinewave and that the last cycle is always completed.

Sample ASK Data Array

0 1 1 1 0 1 0 0 0 1

Parameters

Name	Type	Description
<ask_data>	ASCII	Block of ASCII data that contains information for the generator when to shift from base to shifted amplitude and vice-versa.

PSK Modulation Programming

Use the following commands for programming the PSK parameters. The PSK function can shift from start to shifted phase setting, within the range of 0 to 360°, at a frequency determined by the rate value and controlled by a sequence of bits in the PSK data table. The commands for programming the phase shift keying function are described below. Note that the carrier waveform frequency (CW) setting is common to all modulation schemes.

PSK:PHASe <start_phase>(?)

Description

This programs the start phase of the carrier waveform. The start phase shifts when the pointer in the data array points to “0”.

Parameters

Name	Range	Type	Default	Description
<start_phase>	0 to 360	Numeric	0	Programs the start phase for the carrier waveform in units of degrees.

Response

The 3172 returns the present start phase value.

PSK:PHASe:SHIFted <shift_phase>(?)

Description

This programs the shifted phase. The phase shifts when the pointer in the data array points to “1”.

Parameters

Name	Range	Type	Default	Description
<shift_phase>	0 to 360	Numeric	180	Programs the shift phase for the carrier waveform in units of degrees.

Response

The 3172 returns the present shift phase value.

PSK:RATE <rate>(?)

Description

This allows the user to select PSK word rate. The word rate is the interval of which the bit streams in the PSK data array are clocked, causing the output phase to hop from start to shifted phase values and vice-versa. Note that this command is dedicated for programming the PSK modulation function only.

Parameters

Name	Range	Type	Default	Description
<baud>	1 to 10e6	Numeric	10e3	Programs the rate of which the phase shifts from start to shifted frequency in units of Hz.

Response

The 3172 returns the present baud value. The returned value will be in standard scientific format (for example: 100mHz would be returned as 100e-3 – positive numbers are unsigned).

PSK:DATA <psk_data>

Description

Loads the data stream that will cause the 3172 to hop from phase to phase. Data format is a string of "0" and "1" which define when the output generates the various phases. The size of the data word depends on the PSK function.

Below you can see how a PSK data table is constructed. The PSK data table sample below shows a list of 10 shifts. The 3172 will step through this list, outputting either start or shifted phases, depending on the data list: Zero will generate start phase and One will generate shifted phase. Note that the output waveform is always sinewave and that the last cycle is always completed.

Sample PSK Data Array

0 1 1 1 0 1 0 0 0 1

Parameters

Name	Type	Description
<psk_data>	ASCII	Block of ASCII data that contains information for the generator when to step from one phase setting to another.

PSK:MARKer <index>(?)

Description

Programs where on the data stream the 3172 will generate a pulse, designated as PSK marker, or index point. The marker pulse is generated at the SYNC output connector. Note that if you intend to program marker position, you must do it before you load the PSK data list. The PSK:MARK command is common to all PSK modulation functions.

Parameters

Name	Range	Type	Default	Description
<index>	1 to 4000	Numeric (integer only)	1	Programs a marker pulse at an index bit position.

Response

The 3172 returns the present marker position.

Frequency Hopping Modulation Programming

Use the following commands for programming the frequency hop parameters. Hop control is internal. The frequency will hop from frequency to frequency at a rate determined by the dwell time value and controlled by a sequence of frequencies in the HOP data table.

There are two hop modes: Fixed Dwell, where the rate of which the generator hops from frequency to frequency is constant and Variable Dwell, where the rate of which the generator hops from frequency to frequency is programmable for each hop.

The commands for programming the frequency hopping function are described below. Note that the carrier waveform frequency (CW) setting is common to all modulation schemes.

FHOP:DWELl:MODe {FIXed|VARiable}(?)

Description

This selects between fixed or variable dwell-time for the frequency hops. Select the fixed option if you want each frequency to dwell equally on each step. The variable option lets you program different dwell times for each frequency hop. The 3172 output hops from one frequency to the next according to a sequence given in a hop table. The variable dwell time table contains dwell time data for each step however, the fixed dwell time table does not contain any dwell time information and therefore, if you select the fixed option, make sure your dwell time is programmed as required.

Parameters

Name	Type	Default	Description
FIXed	Discrete	FIX	Selects the fixed dwell time frequency hops mode
VARiable	Discrete		Select the variable dwell time frequency hops mode

Response

The 3172 returns FIX, or VAR depending on the selected dwell setting.

FHOP:DWELl <dwell_time>(?)

Description

This selects the dwell time for frequency hops when the selected mode is Fixed dwell time hops. The dwell time table in this case does not contain the dwell time per step parameters and therefore, the value which is programmed with this command remains constant for the entire hop sequence.

Parameters

Name	Range	Type	Default	Description
<dwell_time>	200e-9 to 20	Numeric	200e-9	Programs dwell time for the fixed dwell-time frequency hop function. The same dwell time will be valid for each frequency hop. Dwell time is programmed in units of s.

Response

The 3172 returns the present dwell time value. The returned value will be in standard scientific format (for example: 100mHz would be returned as 100e-3 – positive numbers are unsigned).

FHOP:FIX:DATA <fix_hop_data>

Description

This command will download the data array that will cause the instrument to hop through the frequency list. The dwell time for each frequency list item is fixed and can be programmed using the HOP:DWEL command. Note that if you intend to program marker position, you must do it first and then load the frequency hops list.

Below you can see how a hop table is constructed. The file sample below shows a list of 10 frequencies. The 3172 will hop through this list, outputting the next frequency each time it hops. Note that the carrier waveform is always sinewave and that the last cycle is always completed even if the dwell time is shorter than the period of the waveform. For example, if you program dwell time of 1ms and the frequency step has frequency of 1Hz (1s period), the frequency step will last 1 second although the dwell time is 1ms.

Sample Frequency Hops Data Array

1e+6 2e+6 3e+3 4e+6 5e+5 6e+2 7e+1 8e+6 9e+3 10e+5

Parameters

Name	Type	Description
<fix_hop_data>	Double	Block of binary data that contains information of frequency values.

FHOP:FIX:DATA <var_hop_data>

Description

This command will download the data array that will cause the instrument to hop through the frequency list. The dwell time for each frequency list item is variable and is supplied in the variable hop table data array. Note that the HOP:DWEL command has no effect on this sequence. Also note that if you intend to program marker position, you must do it first and then load the frequency hops list.

Below you can see how a hop table is constructed. The file sample below shows a list of 10 frequencies and their associated dwell times. The 3172 will hop through this list, outputting the next frequency each time it hops. Note that the carrier waveform is always sinewave and that the last cycle is always completed even if the dwell time is shorter than the period of the waveform. For example, if you program dwell time of 1ms and the frequency step has frequency of 1Hz (1s period), the frequency step will last 1 second although the dwell time is 1ms.

Sample Frequency Hops Data Array

1e+6 100 2e+6 2000 3e+3 3e4 4e+6 40 5e+5 5e3 6e+2 6000 7e+1 0.7 8e+8e2 6 9e+3 90 10e+51000

In the above example, the first number is the frequency value and the second number is its dwell time. Therefore, only even number of sets can be located in this table.

Parameters

Name	Type	Description
<var_hop_data>	Double	Block of binary data that contains information of frequency hop values and their respective dwell time.

FHOP:MARKer <index>(?)

Description

Programs where on the frequency list the 3172 will generate a pulse, designated as Hop marker, or index point. The marker pulse is generated at the SYNC output connector.

Parameters

Name	Range	Type	Default	Description
<index>	1 to 5000	Numeric (integer only)	1	Programs a marker pulse at an index frequency hop position.

Response

The 3172 returns the present marker position.

Amplitude Hopping Modulation Programming

Use the following commands for programming the amplitude hop parameters. Hop control is internal. The amplitude will hop from amplitude level to amplitude level at a rate determined by the dwell time value and controlled by a sequence of amplitudes in the HOP data table.

There are two hop modes: Fixed Dwell, where the rate of which the generator hops from amplitude level to amplitude level is constant and Variable Dwell, where the rate of which the generator hops from amplitude level to amplitude level is programmable for each hop.

The commands for programming the amplitude hopping function are described below. Note that the carrier waveform frequency (CW) setting is common to all modulation schemes.

AHOP:DWELl:MODe {FIXed|VARiable}(?)

Description

This selects between fixed or variable dwell-time for the amplitude hops. Select the fixed option if you want each amplitude level to dwell equally on each step. The variable option lets you program different dwell times for each amplitude hop. The 3172 output hops from one amplitude level to the next according to a sequence given in a hop table. The variable dwell time table contains dwell time data for each step however, the fixed dwell time table does not contain any dwell time information and therefore, if you select the fixed option, make sure your dwell time is programmed as required.

Parameters

Name	Type	Default	Description
FIXed	Discrete	FIX	Selects the fixed dwell time amplitude hops mode
VARiable	Discrete		Select the variable dwell time amplitude hops mode

Response

The 3172 returns FIX, or VAR depending on the selected dwell setting.

AHOP:DWELl <dwell_time>(?)

Description

This selects the dwell time for amplitude hops when the selected mode is Fixed dwell time hops. The dwell time table in this case does not contain the dwell time per step parameters and therefore, the value which is programmed with this command remains constant for the entire hop sequence.

Parameters

Name	Range	Type	Default	Description
<dwell_time>	200e-9 to 20	Numeric	200e-9	Programs dwell time for the fixed dwell-time amplitude hop function. The same dwell time will be valid for each amplitude hop. Dwell time is programmed in units of s.

Response

The 3172 returns the present dwell time value. The returned value will be in standard scientific format (for example: 100mHz would be returned as 100e-3 – positive numbers are unsigned).

AHOP:FIX:DATA <fix_hop_data>

Description

This command will download the data array that will cause the instrument to hop through the amplitude list. The dwell time for each amplitude list item is fixed and can be programmed using the HOP:DWEL command. Note that if you intend to program marker position, you must do it first and then load the amplitude hops list.

Below you can see how a hop table is constructed. The file sample below shows a list of 10 amplitudes. The 3172 will hop through this list, outputting the next amplitude each time it hops. Note that the carrier waveform is always sinewave and that the last cycle is always completed even if the dwell time is shorter than the period of the waveform. For example, if you program dwell time of 1ms and the amplitude step has frequency of 1Hz (1s period), the frequency step will last 1 second although the dwell time is 1ms.

Sample Amplitude Hop Data Array

0 1e0 2e0 3e0 4e+0 5e+0 100e-3 200e-3 300e-3 400e-3 500e-3

Parameters

Name	Type	Description
<fix_hop_data>	Double	Block of binary data that contains information of amplitude values.

AHOP:FIX:DATA <var_hop_data>

Description

This command will download the data array that will cause the instrument to hop through the amplitude list. The dwell time for each amplitude list item is variable and is supplied in the variable hop table data array. Note that the HOP:DWEL command has no effect on this sequence. Also note that if you intend to program marker position, you must do it first and then load the amplitude hops list.

Below you can see how a hop table is constructed. The file sample below shows a list of 10 amplitudes and their associated dwell times. The 3172 will hop through this list, outputting the next amplitude each time it hops. Note that the carrier waveform is always sinewave and that the last cycle is always completed even if the dwell time is shorter than the period of the waveform. For example, if you program dwell time of 1ms and the amplitude step has frequency of 1Hz (1s period), the amplitude step will last 1 second although the dwell time is 1ms.

Sample Amplitude Hop Data Array

1e0 100 2e0 2000 3e0 3000 4e0 4000 5e0 5000 6e0 6000 7e0 7000 8e0 8000 9e0 9000 10e0 10000

In the above example, the first number is the amplitude value and the second number is its dwell time. Therefore, only even number of sets can be located in this table.

Parameters

Name	Type	Description
<var_hop_data>	Double	Block of binary data that contains information of amplitude hop values and their respective dwell time.

AHOP:MARKer <index>(?)

Description

Programs where on the amplitude list the 3172 will generate a pulse, designated as Hop marker, or index point. The marker pulse is generated at the SYNC output connector.

Parameters

Name	Range	Type	Default	Description
<index>	1 to 5000	Numeric (integer only)	1	Programs a marker pulse at an index amplitude hop position.

Response

The 3172 returns the present marker position.

Pulse Waveform Commands (P2 Module Only)

Use the following command for programming the pulse generator functions and their associated parameters. Ignore this section of the manual if you do not have the P2 module installed in your 3172 carrier. There are two independent pulse generator channels in each P2 module; use the command inst:sel to control individual channels. In case you have 3172-W6P2, the pulse channels are designated as 2 and 3 but for the 3172-P2P2 (four channels), the channels are designated as 1, 2, 3, and 4. Figure 5-1 shows the channel designation for the various models.

Table 5-11, Pulse Waveform Command Summary

Keyword	Parameter Form	Default	Association
[:SOURce]			
:PULSe			
:MODe	NORMal | DELayed | DOUBle | HOLDdcycle | EWIDth	NORM	P2
:PERiod	20e-9 to 10	1e-3	P2
:WIDTh	7e-9 to 10	100e-6	P2
:DCYCle	1 to 99	50	P2
:DELay	0 to 10	0	P2
:DOUBle			
:DELay	0 to 10	200e-6	P2
:POLarity	NORMal | COMPlement | INVerted	NORM	P2
:TRANsition			
:STATe	FAST | LINear | SYMMetrical	FAST	P2
[:LEADing]	5e-9 to 5e-3	10e-6	P2
:TRAiling	5e-9 to 5e-3	10e-6	P2

PULSe:MODe {NORMal|DELayed|DOUBle|HOLDdcycle|EWIDth}(?)

Description

This command will program the mode of the pulse. Pulse mode options are: Single pulse, Delayed pulse, Double pulse, Hold duty cycle pulse, External width pulse and PWM1.

Parameters

Name	Type	Default	Description
NORMal	Discrete	NORM	Selects the normal pulse output. In triggered run mode, this selection generates a single pulse with each valid trigger event.

DELayed	Discrete	Selects a delayed pulse mode. Normal pulses are delayed from the SYNC output.
DOUBle	Discrete	Selects a double pulse mode, which generates a pair of single pulses that are displaced by the double delay period value.
HOLDdcycle	Discrete	Programs a pulse mode, which generates a normal pulse that has a fixed duty cycle regardless of the period setting
EWIDth	Discrete	Programs a pulse mode that reconstructs the pulse shape from an external input (TRIG IN)

Response
The 3172 will return SING, DEL, DOUB, HOLD, or EWID depending on the present pulse mode setting.

PULSe:PERiod <period>(?)

Description
This command will program the pulse repetition rate (period). Note that the sum of all parameters, including the pulse width, rise and fall times cannot exceed the programmed pulse period and therefore, it is recommended that the pulse period be programmed first before all other pulse parameters. Note that by selecting the double pulse mode, the pulse period remains unchanged.

Parameters

Name	Range	Type	Default	Description
<period>	20e-9 to 10	Numeric	1e-3	Will program the period of the pulse waveform in units of seconds.

Response
The 3172 will return the present pulse period value in units of seconds.

PULSe:WIDth <width>(?)

Description
This command will program the pulse width value. Note that the only case where the pulse width can exceed the value of the period setting is in triggered mode, where external trigger events determine the period of the pulse.

Parameters

Name	Range	Type	Default	Description
<width>	7e-9 to 10	Numeric	1e-3	Will set the width of pulse in units of seconds. Note that the sum of all parameters, including the pulse width must not exceed the programmed pulse period and therefore, it is recommended that the pulse period be programmed before all other pulse parameters.

Response
The 3172 will return the present pulse width value in units of seconds.

PULSe:DCYCle <duty_cycle>(?)

Description

This command affects the output only when the 3172 is placed in Hold Duty Cycle pulse mode. The programmed duty cycle parameter holds maintains constant duty cycle scenario regardless of the period setting.

Parameters

Name	Range	Type	Default	Description
<duty_cycle>	1 to 99	Numeric	50	Will set the pulse duty cycle in units of percent. Note that this parameter will affect the pulse output only in the Hold Duty Cycle pulse mode.

Response

The 3172 will return the present duty cycle value in units of percent

PULSe:DELay <delay>(?)

Description

This command will program the delayed interval of which the output idles on the low level amplitude until the first transition to high level amplitude. The delay is measured from the SYNC position to the first pulse transition. Note that this delay does not include the system delay error that is specified in Appendix A. Also note that the only case where the delay can exceed the value of the period setting is in triggered mode, where external trigger events determine the period of the pulse.

Parameters

Name	Range	Type	Default	Description
<delay>	0 to 10	Numeric	0	Will set the delay time interval in units of seconds. Delay is measured from the SYNC to the first pulse transition. System delay error is not included in the delay value and must be taken into consideration.

Response

The 3172 will return the pulse delay value in units of seconds.

PULSe:DOUBle:DELay <d_delay>(?)

Description

This command will program the delay between two adjacent pulses when the double mode is selected. Otherwise, the double pulse delay has no effect on the pulse structure. Note that the only case where the delay can exceed the value of the period setting is in triggered mode, where external trigger events determine the period of the pulse.

Parameters

Name	Range	Type	Default	Description
<d_delay>	0 to 10	Numeric	200e-6	Will set the delay between two adjacent pulses for the double pulse mode in units of seconds. Note that the sum of all parameters, including the pulse delay time must not exceed the programmed pulse period and

therefore, it is recommended that the pulse period be programmed before all other pulse parameters.

Response

The 3172 will return the present double pulse delay value in units of seconds.

PULSe:POLarity {NORMal|COMPlemented|INVerted} (?)

Description

This command will program the polarity of the pulse in reference to the base line level. The polarity options are: Normal, where the pulse is generated exactly as programmed; Inverted, where the pulse is inverted about the 0 level base line; and Complemented, where the pulse is inverted about its mid amplitude axis.

Parameters

Name	Type	Default	Description
NORMal	Discrete	NORM	Programs normal pulse output
COMPlemented	Discrete		Programs complemented pulse output
INVerted	Discrete		Programs an inverted pulse output

Response

The 3172 will return NORM, COMP or INV depending on the present polarity setting.

PULSe:TRANsition:STATe {FAST|LINear|SYMMetrical} (?)

Description

This command will program select of the leading edges will transition linearly or in the fastest way. The transition options are: Fast and Linear.

Parameters

Name	Type	Default	Description
FAST	Discrete	FAST	Programs the fast transitions mode. In this mode the leading and trailing edges will transition as fast as the instrument allows and as specified in Appendix A.
LINear	Discrete		Programs the linear transitions. The transitions are allowed within 6 ranges, where the leading edge setting sets the operational range and the leading transition must be programmed within the same range. Additional information on the range settings is provided in Chapter 3 of this manual
SYMMetrical	Discrete		Programs a special mode where the transitions are symmetrical for both the leading and trailing edges, regardless if you program the leading or the trailing edge parameter, the other parameter will automatically be adjusted to have the same value

Response

The 3172 will return FAST, LIN or SYMM depending on the present transition setting.

PULSe:TRANsition <leading_edge>(?)

Description
This command will program the interval it will take the leading edge of the pulse to transition from its low to high level settings. The parameter is programmed in units of seconds. Transition times are programmed within 6 ranges of which both leading and trailing edges must reside in the same range. The leading edge setting determines the range. More information on this function is available in Chapter 3. Note that this parameter will affect the instrument only when the pulse transition mode is set to linear.

Parameters

Name	Range	Type	Default	Description
<leading_edge>	5e-9 to 5e-3	Numeric	10e-6	Will set the leading edge transition time parameter in units of seconds. Note that the sum of all parameters, including transition times must not exceed the programmed pulse period and therefore, it is recommended that the pulse period be programmed before all other pulse parameters.

Response
The 3172 will return the present leading edge transition time value in units of seconds.

PULSe:TRANsition:TRAiling <fall>(?)

Description
This command will program the interval it will take the trailing edge of the pulse to transition from its high to low level settings. The parameter is programmed in units of seconds. Transition times are programmed within 6 ranges of which both leading and trailing edges must reside in the same range. The leading edge setting determines the range. More information on this function is available in Chapter 3.Note that this parameter will affect the instrument only when the pulse transition mode is set to linear.

Parameters

Name	Range	Type	Default	Description
<fall>	5e-9 to 5e-3	Numeric	10e-6	Will set the fall time parameter. Note that the sum of all parameters, including the fall time must not exceed the programmed pulse period and therefore, it is recommended that the pulse period be programmed before all other pulse parameters.

Response
The 3172 will return the present trailing edge transition time value in units of seconds.

Half Cycle Control Commands (W6 Module Only)

Use the following commands for programming the half cycle functions and their associated parameters. There are three half cycle functions: Sine, Triangle and Square. The specifications and limitations of the half cycle functions are specified in Appendix A. The half cycle commands are summarized in Table 5-12.

Table 5-12, Half Cycle Command Summary

Keyword	Parameter Form	Default	Association
:HALFcycle			
:DELay	200e-9 to 20	1e-6	W6
:DCYCle	0 to 99.99	50	W6
:FREQuency	10e-3 to 1e6	1e6	W6
:PHASe	0 to 360	0	W6
:SHAPe	SINusoid | TRIangle | SQUare	SIN	W6

HALFcycle:DELay <delay>(?)

Description

This command programs the interval of which the output idles between half cycles. The idle level is normally 0 V except if programmed otherwise with the volt:offs command.

Parameters

Name	Range	Type	Default	Description
<delay>	200e-9 to 20	Numeric	1e-6	Sets the delay time interval between half cycles in units of seconds.

Response

The 3172 returns the half cycle delay value in units of seconds.

HALFcycle:DCYCle <duty_cycle>(?)

Description

This command programs the duty cycle of the square waveform when the half cycle square shape is selected. Note that this command has no effect on the standard square wave duty cycle.

Parameters

Name	Range	Type	Default	Description
<duty_cycle>	0 to 99.99	Numeric	50	Sets the delay time interval between half cycles in units of seconds.

Response

The 3172 returns the square wave duty cycle value in units of percent.

HALFcycle:FREQuency <freq>(?)

Description

This command programs the frequency of the half cycle waveforms in units of hertz (Hz). It has no effect on the frequency of other waveform functions.

Parameters

Name	Range	Type	Default	Description
<freq>	10e-3 to 1e6	Numeric	1e6	Sets the frequency of the half cycle waveform in units of Hz. This parameter does not affect the frequency of other waveform functions.

Response

The 3172 returns the present half cycle frequency value. The returned value will be in standard scientific format (for example: 100mHz would be returned as 100e-3 – positive numbers are unsigned).

HALFcycle:PHASe <phase>(?)

Description

This command programs the start phase of the half cycle sine and triangle waveform. This command has no effect on other waveform functions.

Parameters

Name	Range	Type	Default	Description
<phase>	0 to 360	Numeric	0	Programs the start phase parameter for the half cycle sine and triangle waveforms in units of degrees. The phase can be programmable with resolution of 0.05° throughout the entire frequency range of the half cycle function.

Response

The 3172 returns the present start phase value.

HALFcycle:SHAPe {SINusoid|TRIangle|SQUare}(?)

Description

This command defines the type of half cycle waveform that will be available at the output connector.

Parameters

Name	Type	Default	Description
SINusoid	Discrete	SIN	Selects the half cycle sine waveform.
TRIangle	Discrete		Selects the half cycle triangular waveform.
SQUare	Discrete		Selects the half cycle square waveform.

Response

The 3172 returns SIN, TRI, or SQU depending on the present 3172 setting

Counter Control Commands (W6 Module Only)

Use the following commands for programming the counter/timer measuring function and its associated parameters. The counter/timer function is created digitally, however, it closely simulates a stand-alone instrument so its functions are programmed just as they would be programmed on a dedicated instrument. The specifications and limitations of the counter/timer are specified in Appendix A. The counter commands are summarized in Table 5-13.

Table 5-13, Counter Command Summary

Keyword	Parameter Form	Default	Association
:COUNter			
:FUNCtion	FREQuency | PERiod | APERiod | PULSe | TOTalize	FREQ	W6
:DISPlay			
:MODE	NORMal | HOLD	NORM	W6
:GATE			
:TIME	100e-6 to 1	1	W6
:RESet			W6
:READ			W6

COUNter:FUNCtion {FREQuency|PERiod|APERiod|PULSe|ITOTalize| GTOTalize(?)

Description

This command programs the measurement function for the counter/timer. Each measurement can be set up with its gate time (where applicable) and display mode.

Parameters

Name	Type	Default	Description
FREQuency	Discrete	FREQ	Selects the frequency measurement function. Frequency is measured on continuous signal only. The result of the frequency measurement has gate-dependent resolution. The 3172 displays 7 digits of frequency reading in one second of gate time. If the gate time is decreased, the number of displayed digits decreases proportionally to the gate time interval. Reduce the gate time when you want to accelerate the reading process however, always make sure that the period of the signal is smaller than the gate time setting.
PERiod	Discrete		Selects the period measurement function. Period can be measured on either continuous or non-repetitive signals. Since the period of the signal is directly proportional to the gating time, the number of displayed digits decreases proportionally to the period of the signal. If you need to have more resolution and you signal is repetitive, use the period averaged measurement function. The best resolution in period measurements is 100 ns.
APERiod	Discrete		Selects the period averaged measurement function. Period

			averaged can be measured continuous signals only. In fact, this is the inverse function of frequency and therefore, gate time determines the resolution of the reading. Reduce the gate time when you want to accelerate the reading process however, always make sure that the period of the signal is smaller than the gate time setting.
PULSe	Discrete		Selects the pulse width measurement function. Pulse width can be measured on either continuous or non-repetitive signals. Since the width of the signal is directly proportional to the gating time, the number of displayed digits decreases proportionally to the pulse width of the signal. The best resolution in period measurements is 10 ns.
ITOTalize	Discrete		Selects the totalize measurement function. In this mode, the gate opens when the first valid signal is sensed at the counter input and remains open until programmed otherwise. Pulse are counted and displayed continuously until intervened externally. The counter can accumulate 8 digits before it will overflow. An overflow indication is available.

Response

The 3172 returns FREQ, PER, APER, PULS, or ITOT depending on the present measurement function setting.

COUNter:DISPlay:MODe {NORMal|HOLD(?)

Description

This command programs the display time mode for the counter/timer. The two modes are normal for continuous display readings and hold for single reading after arming the counter input.

Parameters

Name	Type	Default	Description
NORMal	Discrete	NORM	Selects the continuous reading mode. In this case, the counter input is self-armed, which means that every valid signal that is sensed at the trigger input connector will be counted and measured processed and results placed on the interface port.
HOLD	Discrete		Selects the single reading mode. In this case, the counter input is armed first and the first valid signal that is sensed at the trigger input connector will be counted and measured and its result processed and placed on the interface port.

Response

The 3172 returns NORM, INV or COMP depending on the present polarity setting

COUNter:GATE <time>(?)

Description

This command programs the gate time interval for frequency, period averaged and totalize in gated mode. Measurements will be taken only after the input has been armed and valid signal available at the input connector. Notice however, that the gate time internal must be larger than the period of the measure signal.

Parameters

Name	Range	Type	Default	Description
<time>	100e-6 to 1	Numeric	1	Programs the gate time interval in units of seconds. In continuous mode, the counter is self-armed and therefore every valid signal at the counter input will open the gate and initiate a measurement cycle. In hold mode, the counter must be armed before the gate can open. Always make sure the programmed gate time interval is larger than the period of the measured signal.

Response

The 3172 returns the present gate time value in units of seconds.

COUNter:RESet

Description

This command resets the counter/timer and arm the instrument for its next reading.

COUNter:READ

Description

This command interrogates the counter/timer for a reading. Note that the read command must follow a valid gate time interval otherwise reading will not be available and the interface bus will be held until the measurement cycle has been completed and result available to be read.

Response

The 3172 returns the result of the present measurement function reading. The returned value will be in standard scientific format (for example: 10 MHz would be returned as 10e6 – positive numbers are unsigned).

Power Amplifier Control Commands (A3 Module Only)

Use the following commands for programming the power amplifier module. The power amplifier module does not generate signals of its own, but receives signals from the W6 waveform generator module. The specifications and limitations of the power amplifier timer are provided in Appendix A. Power amplifier commands are summarized in Table 5-15.

Table 5-15, Power Amplifier Command Summary

Keyword	Parameter Form	Default	Association
:OUTPut			
:AMPLifier	AUTO | HIGH | LOW	AUTO	A3
:POWer	OFF | ON | 0 | 1	0	A3

OUTPut:AMPLifier {AUTO|HIGH|LOW}(?)

Description

This command selects the internal voltage levels of the power amplifier supply rails. Setting the correct levels is important to avoid exceeding the temperature limits of the power amplifier component.

Parameters

Name	Type	Default	Description
AUTO	Discrete	AUTO	Selects automatic selection of the supply rail voltages for best power efficiency.
HIGH	Discrete		Use this option for maximum output levels at the output connector. This option is recommended for an output range of 61 Vp-p to 122 Vp-p
LOW	Discrete		Use this option for low output levels at the output connector. This option is recommended for an output range of 20 Vp-p to 61 Vp-p.

Response

The 3172 returns AUTO, HIGH, or LOW depending on the present voltage setting.

OUTPut:AMPLifier:POWer{OFF|ON|0|1}(?)

Description

This parameter sets the power amplifier circuit to the ON (amplifier enabled) or OFF (bypassed) condition. Note that in order to have an output signal at the output connector requires that both this command be turned on and the protection switch set at its on position.

Parameters

Name	Type	Default	Description
0-1	Discrete	0	Turns the output amplifier circuits on and off.

Response

The 3172 returns 0 or 1 depending on the selected option.

Synchronization Commands (W6 Modules Only)

Synchronization commands control phase locking to an external source. The source is applied via the front panel TRIG/PLL IN connector, or from another module in the same VXI chassis through the Local Bus, TTL Trigger Bus, or ECL Trigger bus.

Note that synchronization through the VXI Local Bus (LBUS0-7) is available only in Modern Mode, and does not use the legacy commands discussed in this section. To set up and control Local Bus synchronization, refer to the INSTrument:COUPle commands.

When locking the 3172 to an external source applied to the TRIG/PLL IN connector, the auto-detection circuit sets up the lock-in range and prepares the PLL circuits for a final phase lock. After phase lock has been achieved, the 3172 duplicates the frequency and start phase of the external source. A front-panel LED illuminates when the external source signal is valid and the instrument has locked onto it.

When phase locking to a signal on the TRIG/PLL IN connector, phase adjustment commands allow phase shifting of the locked signal from -180° to +180° referenced to the input signal. In addition, the phase can be shifted by applying a signal to the front-panel PM IN connector, where DC to 10 KHz voltage levels can phase-modulate the locked instrument.

The ECLTrig bus in the VXI chassis allows synchronizing multiple 3172s inside the same VXIbus chassis without external connections. The ECLT0 and ECLT1 lines provide the necessary signals to achieve lock between modules. Using this method, one instrument is configured as master and the rest of the instruments are configured as slaves. They do not need to be in adjacent chassis slots. When synchronized, the slave instruments are initially locked to the start phase of the master but later can be configured to have phase offsets relative to the master. Phase offset is programmable from 0° to 360°.

When synchronizing through the ECLTrig bus or TTLTrig bus, phase offset is programmable from 0° to 360°. Phase offset resolution depends on the number of waveform samples. For instance, if there are 1,000 waveform samples, the phase offset is adjustable in 0.36° increments, since 360° / 1,000 samples = 0.36° per sample.

As another example, if there are only ten waveform samples, the smallest phase offset increment is 360° / 10 samples, or 36°.

The phase offset discussed above is known as the coarse phase offset. To improve the resolution for waveforms having fewer samples, a separate fine phase adjustment takes the resolution down to 0.01°. The fine phase adjustment and can be used in conjunction with any valid coarse phase offset setting.

Synchronization commands are summarized in Table 5-14.

Table 5-14, Synchronization Command Summary

Keyword	Parameter Form	Default	Association
[:SOURce]			
:PHASe1			
:LOCK			W6
[:STATe]	OFF | ON | 0 | 1	0	W6
:ADJust	0 to 360	0	W6
:SOURce	MASTer | SLAVe	MAST	W6
:PHASe2		(=PLL)	W6
:LOCK			
[:STATe]	OFF | ON | 0 | 1	0	W6
:SOURce	EXTernal | TTLTrg<n> | ECLTrg 0	EXT	W6
:ADJust	-180 to 180	0	W6
:FINe	-36 to 36	0	W6

PHASe1:LOCK {OFF|ON|0|1}(?)

Description

This command turns the backplane synchronization sequence on and off. This command requires that another 3172 is plugged in the same chassis. The location of the two instruments is not critical for them to synchronize. Using this synchronization method, the sample clock is placed on the ECLTrg0 line and the synchronization signal is place on ECLTrg0 and therefore, if you intend to use this synchronization method, make sure that these two backplane lines are not used by other modules.

Parameters

Range	Type	Default	Description
0-1	Discrete	0	Turns the backplane synchronization on and off

Response

The 3172 returns 0, or 1 depending on the present backplane synchronization setting.

PHASe1:ADJust <phase>(?)

Description

This command programs the phase offset between the master and the slave units where the master is the reference waveform and the phase of the slave instruments are shifted in reference to the master instrument.

Parameters

Name	Range	Type	Default	Description
<phase>	0 to 360	Numeric	0	Programs the phase offset between the slave and the master instrument. The phase is programmed in units of degrees (°). Note however, that the phase offset resolution depends on the number of points that create the waveform. For example, waveform that is made of 1024 points can be shifted with increments of 0.35°,

but another waveform that has only 100 points can be shifted with increments of 3.6°.

Response

The 3172 returns the present phase offset value in units of degrees.

PHASe1:LOCK {MASTer|SLAVe}(?)

Description

By definition, all 3172 units are turned on as masters. This does not interfere with normal operation because the electrical circuits are designed to handle shared nodes. This means that for synchronization purpose, slave units must be programmed to slave state. The sequence to synchronize then is: first, determine who is master and who is slave using this command and then, switch synchronization on using the phas:lock 1 command.

Parameters

Name	Type	Default	Description
MASTer	Discrete	MAST	Selects the master unit in a multi-instrument system. The master feeds the sample clock and the synchronization signals through the ECLTrg 0-1 lines.
SLAVe	Discrete		Selects the slave unit(s) in a multi-instrument system. The slave instruments receive the sample clock and the synchronization signals from the ECLTrg 0-1 lines.

Response

The 3172 returns MAST, or SLAV depending on the present backplane synchronization setting.

PHASe2:LOCK {OFF|ON|0|1}(?)

Description

This command turns the PLL (phase lock loop) function on and off. The reference signal is applied to a front panel input (PLL IN) and the 3172 locks onto it automatically using a smart frequency/phase sensing sequence. After lock, the phase of the 3172 can be shifted in reference to the input signal. The PLL operates in standard and arbitrary waveform modes, locking to external signals ranging from, as low as, 100 Hz to over 10 MHz. When placed in PLL mode, the LED near its input connector blinks until achieving full lock. When locked, the LED lights constantly as an indication that the signal is locked to an external reference.

Parameters

Range	Type	Default	Description
0-1	Discrete	0	Turns the PLL function on and off

Response

The 3172 returns 0, or 1 depending on the present PLL function setting.

PHASe2:LOCK:SOURce {EXTernal|TTLTrg<n>|ECLTrg0}(?)

Description

This command selects the source for the PLL function. Source options are: External signal, applied to the front panel PLL IN connector; Trigger signals, applied through the backplane trigger lines; or, trigger signal applied through the ECLT0 line. While the last two options have fixed logic levels, the front panel input may accept various levels, including such levels that reside on offsets.

Parameters

Name	Type	Default	Description
EXTernal	Discrete	EXT	Selects the front panel PLL IN input as the reference signal. The input can be programmed for trigger level, to adjust the threshold and for slope, to define the reference slope.
TTLTrg<n>	Discrete		Selects one of the backplane trigger lines (TTLTrg 0 through 7) as the reference input.
ECLTrg<n>	Discrete		Selects the backplane ECLTrg0 line as the reference input.

Response

The 3172 returns EXT, TTLT<n>, or ECLT0 depending on the selected PLL reference source setting.

PHASe2:ADJust <phase>(?)

Description

This command programs the phase offset between the reference input and the 3172 output. The output start phase is shifted in reference to the external signal.

Parameters

Name	Range	Type	Default	Description
<phase>	-180 to 180	Numeric	0	Programs the phase offset between the reference and the output waveform. The phase is programmed in units of degrees (°). Note however, that the phase offset resolution depends on the number of points that create the waveform. For example, waveform that is made of 1024 points can be shifted with increments of 0.35°, but another waveform that has only 100 points can be shifted with increments of 3.6°.

Response

The 3172 returns the present phase offset value in units of degrees.

PHASe2:FINE <fine_phase>(?)

Description

This command programs the phase offset between the reference input and the 3172 output in smaller increments. The output start phase is shifted in reference to the external signal.

Parameters

Name	Range	Type	Default	Description
<phase>	-36 to 36	Numeric	0	Programs the phase offset between the reference and the output waveform in smaller increments. The phase is programmed in units of degrees (°). Note however, that the phase offset resolution depends on the number of points that create the waveform. For example, waveform that is made of 1024 points can be shifted with increments of 0.35°, but another waveform that has only 100 points can be shifted with increments of 3.6°.

Response

The 3172 returns the present fine phase offset value in units of degrees.

LAN System Configuration Commands

The LAN system configuration commands are available with the Model 3172 only (Message Based carrier) that has the LAN connector installed on its front panel. Use these commands to configure module address and other LAN parameters. It is strongly recommended that this be done with a computer specialist because wrong programming may place the instrument in an unknown configuration which may lock the LAN operation completely and only hard reset will be necessary to restore the instrument to its original defaults.

NOTE

Last LAN configuration settings will remain as long as the instrument is turned on. New LAN configuration settings will take effect only after the instrument has been powered off and on.

Table 5-15, LAN Command Summary

Keyword	Parameter Form	Default	Association
:SYSTem			
:IP			
[:ADDRess]	<IP_address>		3172
:MASK	<mask>		3172
:GATeway	<gate_way>		3172
:BOOTp	OFF | ON | 0 | 1	0	3172
HOSTname:	<host_name>		3172
:KEEPalive			
:STATe	OFF | ON | 0 | 1	1	3172
:TIMEout	2 to 300	45	3172
:PROBes	2 to 10	2	3172

SYSTem:IP <ip_adrs>(?)

Description

This command programs the IP address for LAN operation. The programming must be performed from either USB or GPIB controllers.

Parameters

Name	Range	Type	Description
<ip_adrs>	0 to 255	String	Programs the IP address for LAN operation. Programming must be performed from USB or GPIB interfaces. Current IP address can be observed on LAN Properties front panel display.

Response

The 3172 returns the present IP address value similar to the following: 192.168.0.6

SYSTem:IP:MASK <mask_adrs>(?)

Description

This command programs the subnet mask address for LAN operation. The programming must be performed from either USB or GPIB controllers.

Parameters

Name	Range	Type	Description
<mask_adrs>	0 to 255	String	Programs the subnet mask address for LAN operation. Programming must be performed from USB or GPIB interfaces. Current subnet mask address can be observed on LAN Properties front panel display.

Response

The 3172 returns the present IP address value similar to the following: 255.255.255.0

SYSTem:IP:BOOTp {OFF|ON|0|1}(?)

Description

Use this command to toggle BOOTp mode on and off.

Parameters

Range	Type	Default	Description
0-1	Discrete	0	Toggles BOOTp mode on and off. When on, the IP address is administrated automatically by the system

Response

The 3172 returns 0, or 1 depending on the present BOOTp setting.

SYSTem:IP:GATeway <gate_adrs>(?)

Description

This command programs the gateway address for LAN operation. The programming must be performed from either USB or GPIB controllers.

Parameters

Name	Range	Type	Description
<gate_adrs>	0 to 255	String	Programs the gateway address for LAN operation. Programming must be performed from USB or GPIB interfaces. Current gateway address can be observed on LAN Properties front panel display.

Response

The 3172 returns the present IP address value similar to the following: 0.0.0.0

SYSTem:IP:HOSTname <name>(?)

Description

This command programs the host name address for LAN operation. The programming is performed in the factory and it is highly suggested that users do not change the host name without first consulting a Tabor customer service person.

Parameters

Name	Type	Description
<name>	String	Programs the host name for LAN operation.

Response

The 3172 returns a string containing the host name. String length is 16 characters.

SYSTem:KEEPalive:STATe {OFF|ON|0|1}(?)

Description

Use this command to toggle the keep alive mode on and off. The keep alive mode assures that LAN connection remains uninterrupted throughout the duration of the LAN interfacing.

Parameters

Range	Type	Default	Description
0-1	Discrete	1	Toggles the keep alive mode on and off. When on, the 2572 constantly checks for smooth LAN connection at intervals programmed by the syst:keep:time command. The LAN will be probed as many times as programmed by syst:keep:prob parameter to check if there is an interruption in the LAN communication. When communication fails, the 3172 reverts automatically to local (front panel) operation.

Response

The 3172 returns 0, or 1 depending on the present keep alive setting.

SYSTem:KEEPalive:TIMEout <time_out>(?)

Description

This command programs the keep alive time out. The keep alive mode assures that LAN connection remains uninterrupted throughout the duration of the LAN interfacing.

Parameters

Name	Range	Type	Default	Description
<time_out>	2 to 300	Numeric	45	Programs the keep alive time out in units of seconds. The time out period is initiated when the LAN is idle for more than the time out period. The LAN will be probed as many times as programmed by syst:keep:prob parameter to check if there is an interruption in the LAN communication. When communication fails, the 3172 reverts automatically to local (front panel) operation.

Response

The 3172 returns the present keep alive time out value.

SYSTem:KEEPalive:PROBes <probes>(?)

Description

This command programs the number of probes that are used by the keep alive sequence. The keep alive mode assures that LAN connection remains uninterrupted throughout the duration of the LAN interfacing.

Parameters

Name	Range	Type	Default	Description
<probes>	2 to 10	Numeric	2	Programs the number of probes that are used by the keep alive sequence. The time out period is initiated when the LAN is idle for more than the time out period and the LAN will be probed as many times as programmed by this parameter to check if there is an interruption in the LAN communication. When communication fails, the 3172 reverts automatically to local (front panel) operation.

Response

The 3172 returns the present keep alive number of probes.

System Commands

The system-related commands are not related directly to waveform generation but are an important part of operating the generator. These commands can reset or test the instrument, or query the instrument for system information.

Table 5-16, System Command Summary

Keyword	Parameter Form	Default	Association
:RESet			W6, P2
:SYSTem			
:ERRor?			W6, P2
:VERSion?			3172
:INFOrmation			
:CALibration?			3172
:MODel?			3172
:SERial?			3172
:SMEMory			
:MODe	READ | WRITe	WRIT	W6
[:STATe]	OFF | ON | 0 | 1	0	W6
:TEST			
[:ALL]?			3172

RESet, or *RST

Description

This command will reset the 3172 to its factory defaults.

SYSTem:ERRor?

Description

Query only. This query will interrogate the 3172 for programming errors.

Response

The 3172 returns error code. Error messages are listed later in this manual.

SYSTem:VERSion?

Description

Query only. This query will interrogate the 3172 for its current firmware version. The firmware version is automatically programmed to a secure location in the flash memory and cannot be modified by the user except when performing firmware update.

Response

The 3172 returns the current firmware version code in a format similar to the following: 1.35

SYSTem:INFormation:CALibration?

Description

Query only. This query will interrogate the instrument for its last calibration date.

Response

The generator returns the last calibration date in a format similar to the following: 24 Oct 2006 (10 characters maximum).

SYSTem:INFormation:MODel?

Description

Query only. This query will interrogate the instrument for its model number in a format similar to the following: 3172. The model number is programmed to a secure location in the flash memory and cannot be modified by the user.

Response

The generator returns its model number: 3172.

SYSTem:INFormation:SERial?

Description

Query only. This query will interrogate the instrument for its serial number. The serial number is programmed to a secure location in the flash memory and cannot be modified by the user.

Response

The generator returns its serial number in a format similar to the following: 000000451 (10 characters maximum).

SMEMory {WRITe|READ }(?)

Description

This command programs read and write switches for the 3172 shared memory block. Shared memory transfer is the fastest way to get waveforms into the 3172. In shared memory mode, the 3172's CPU disconnects from the waveform memory and passes access to the VXIbus. The internal data bus is connected directly to the VXIbus, and data is downloaded into the memory in binary blocks using A32 (or optionally A24) memory space. Byte and bit order are the same as with the Arbitrary Block transfers as described in the Arbitrary Waveform Commands section. After the data is loaded into the instrument, control is returned to the CPU. In shared memory mode, the 3172’s memory acts similar to Direct Memory Access

(DMA). The instrument has to be told when to receive data, send data, surrender or gain control. The 3172 has an auto-increment address counter and therefore, the slot 0 controller must define the base address for both write and read cycles.

Parameters

Name	Type	Default	Description
WRITe	Discrete	EXT	Will prepare the shared memory to accept data from the backplane data bus. Data will not be shared before this function is turned on using smem 1.
READ	Discrete		Will prepare the shared memory to place data on the backplane data bus. Data will not be shared before this function is turned on using smem 1.

Response

The 3172 returns WRIT or READ depending on the selected shared memory setting.

SMEMory {OFF|ON|0|1}(?)

Description

Use this command to toggle the shared memory function on and off. Use smem:mode writ to prepare the instrument for DMA data load from the backplane data bus.

Parameters

Range	Type	Default	Description
0-1	Discrete	1	Toggles the shared memory function on and off. When place in on position, the instrument cannot accept normal data until the data transfer has been completed. Programming examples are given in the driver code.

Response

The 3172 returns 0, or 1 depending on the present shared memory setting.

TEST?

Description

Use this command to test the functionality of the 3172. Bear in mind that this test does not replace the performance checks but comes to provide basic confidence that the instrument operates and responds correctly to basic commands and functions.

Response

The 3172 returns 0 when no error is detected. Non-zero response implies problems in one or more of the tested circuits that requires further investigation by a qualified test engineer.

IEEE-STD-488.2 Common Commands and Queries

Since most instruments and devices in an ATE system use similar commands that perform similar functions, the IEEE-STD-488.2 document specifies a common set of commands and queries that all compatible devices must use. This avoids situations where devices from various manufacturers use different sets of commands to enable functions and report status. The IEEE-STD-488.2 treats common commands and queries as device dependent commands. For example, *TRG is sent over the bus to trigger the instrument. Some common commands and queries are optional, but most of them are mandatory.

Table 5-17, Common Command Summary

Keyword	Parameter Form	Default	Association
*CLS			3172
*ESE	1 to 255	1	3172
*OPC			3172
*RST			3172
*SRE	1 to 255	1	3172
*TRG			3172
*ESE?			3172
*ESR?			3172
*IDN?			3172
*OPC?			3172
*OPT?			3172
*STB?			3172
*TST?			3172

*CLS

Description

Use this command to clear the Status Byte summary register, error register and all event registers. This command has no effect on parameter settings.

*ESE <enable_bits>

Description

Use this command to enable bits in the Standard Event enable register. The selected bits are then reported to the status byte. Information on the standard event register is given in the following.

Parameters

Name	Range	Default	Description
<enable_bits>	0-255	0	Programs the event that will cause the register to report a problem. Setting of “0” disables this feature.

*ESE?

Description

Use this command to query the programmed bits in the Standard Event enable register.

Response

The generator returns a decimal value in the range of 0 to 255, which corresponds to the binary-weighted sum of all bits, set in the register.

*ESR?

Description

Use this command to query the response of the Standard Event enable register. Information on the standard event register is given in the following.

Response

The generator returns a decimal value in the range of 0 to 255, which corresponds to the binary-weighted sum of all bits, set in the register.

*IDN?

Description

Use this command to query the identity of the 3172.

Response

The generator returns data organized into four fields, separated by commas. The generator responds with its manufacturer and model number in the first two fields, and may also report its serial number and options in fields three and four. If the latter information is not available, the device must return an ASCII 0 for each. For example, 3172 response to *IDN? is: Racal Instruments,3172,0,1.0

*OPC

Description

Use this command to set the "operation complete" bit (bit 0) in the Standard Event register after the previous commands have been executed.

*OPC?

Description

Use this command to synchronize between a controller and the instrument using the MAV bit in the Status Byte or a read of the Output Queue. The *OPC? query does not affect the OPC Event bit in the Standard Event Status Register (ESR). Reading the response to the ***OPC?** query has the advantage of removing the complication of dealing with service requests and multiple polls to the instrument. However, both the system bus and the controller handshake are in a temporary hold-off state while the controller is waiting to read the *OPC? query response.

Response

Returns "1" to the output buffer after all the previous commands have been executed.

*OPT?

Description

Use this command to query the options that are installed in this specific module.

Response

Returns "0" if no option is installed or “2” for the 4 Meg memory option.

*RST

Description

Use this command to reset the instrument to its default setting. Factory defaults are listed in the Default column in Table 5-1.

*SRE <enable_bits>

Description

Use this command to enable bits in the Service Request Enable register. The selected bits are then reported to the status byte. Information on the service request register is given in the following.

Parameters

Name	Range	Default	Description
<enable_bits>	0-63, 128-191	0	Programs the event that will cause the register to request service. Setting of “0” disables this feature.

*SRE?

Description

Use this command to query the programmed bits in the Service Request Enable register.

Response

The generator returns a decimal value in the range of 0 to 63 or 128 to 191 since bit 6 (RSQ) cannot be set. The binary-weighted sum of the number represents the value of the bits of the Service Request enable register.

*STB?

Description

Use this command to query the Status Byte for reported errors or events..

Response

The generator returns a summary of the Status Byte register. The *STB? command is similar to a serial poll sequence but is processed like any other instrument command. The *STB? command returns the same result as a serial poll, except the "request service" bit (bit 6) is not cleared if a serial poll has occurred.

*TRG

Description

Use this command from a remote interface as a soft trigger in lieu of an external generator. This command affects the generator if it is first placed in the Trigger or Burst mode of operation and the trigger source is set to "BUS".

*TST?

Description

Use this command to test the functionality of the 3172. Bear in mind that this test does not replace the performance checks but comes to provide basic confidence that the instrument operates and responds correctly to basic commands and functions.

Response

The 3172 returns 0 when no error is detected. Non-zero response implies problems in one or more of the tested circuits that requires further investigation by a qualified test engineer.

The SCPI Status Registers

The 3172 uses the Status Byte register group and the Standard Event register group to record various instrument conditions. Figure 5-6 shows the SCPI status system.

An Event Register is a read-only register that reports defined conditions within the generator. Bits in an event register are latched. When an event bit is set, subsequent state changes are ignored. Bits in an event register are automatically cleared by a query of that register or by sending the *CLS command. The *RST command or device clear does not clear bits in an event register. Querying an event register returns a decimal value, which corresponds to the binary-weighted sum of all bits, set in the register.

An Event Register defines which bits in the corresponding event register are logically ORed together to form a single summary bit. The user can read from and write to an Enable Register. Querying an Enable Register will not clear it. The *CLS command does not clear Enable Registers but it does clear bits in the event registers. To enable bits in an enable register, write a decimal value that corresponds to the binary-weighted sum of the bits required to enable in the register.

The Status Byte Register (STB)

The Status Byte summary register contains conditions from the other registers. Query data waiting in the generator's output buffer is immediately reported through the Message Available bit (bit 4). Bits in the summary register are not latched. Clearing an event register will clear the corresponding bits in the Status Byte summary register. Description of the various bits within the Status Byte summary register is given in the following:

Bit 0 - Decimal value 1. Not used, always set to 0.

Bit 1 - Decimal value 2. Not used, always set to 0.

Bit 2 - Decimal value 4. Not used, always set to 0.

Bit 3 - Decimal value 8. Not used, always set to 0.

Bit 4 - Decimal value 16. Message Available Queue Summary Message (MAV). The state of this bit indicates whether or not the output queue is empty. The MAV summary message is true when the output queue is not empty. This message is used to synchronize information exchange with the controller. The controller can, for example, send a query command to the device and then wait for MAV to become true. If an application program begins a read operation of the output queue without first checking for MAV, all system bus activity is held up until the device responds.

Bit 5 - Decimal value 32. Standard Event Status Bit (ESB) Summary Message. This bit indicates whether or not one or more of the enabled ESB events have occurred since the last reading or clearing of the Standard Event Status Register.

Bit 6 - Decimal value 64. Master Summary Status (MSS)/Request Service (RQS) Bit. This bit indicates if the device has at least one condition to request service. The MSS bit is not part of the IEEE-STD-488.1 status byte and will not be sent in response to a serial poll. However, the RQS bit, if set, will be sent in response to a serial poll.
Bit 7 - Decimal value 128. Not used, always set to 0.

Reading the Status Byte Register

The Status Byte summary register can be read with the *STB? common query. The *STB? common query causes the generator to send the contents of the Status Byte register and the MSS (Master Summary Status) summary message as a single <NR1 Numeric Response Message> element. The response represents the sum of the binary-weighted values of the Status Byte Register. The *STB? common query does not alter the status byte.

Clearing the Status Byte Register

Removing the reasons for service from Auxiliary Status registers can clear the entire Status Byte register. Sending the *CLS command to the device after a SCPI command terminator and before a Query clears the Standard Event Status Register and clears the output queue of any unread messages. With the output queue empty, the MAV summary message is set to FALSE. Methods of clearing other auxiliary status registers are discussed in the following paragraphs.

Service Request Enable Register (SRE)

The Service Request enable register is an 8-bit register that enables corresponding summary messages in the Status Byte Register. Thus, the application programmer can select reasons for the generator to issue a service request by altering the contents of the Service Request Enable Register.

The Service Request Enable Register is read with the *SRE? common query. The response to this query is a number that represents the sum of the binary-weighted value of the Service Request Enable Register. The value of the unused bit 6 is always zero.

The Service Request Enable Register is written using the *SRE command followed by a decimal value representing the bit values of the Register. A bit value of 1 indicates an enabled condition. Consequently, a bit value of zero indicates a disabled condition. The Service Request Enable Register is cleared by sending *SRE0. The generator always ignores the value of bit 6. Summary of *SRE commands is given in the following.

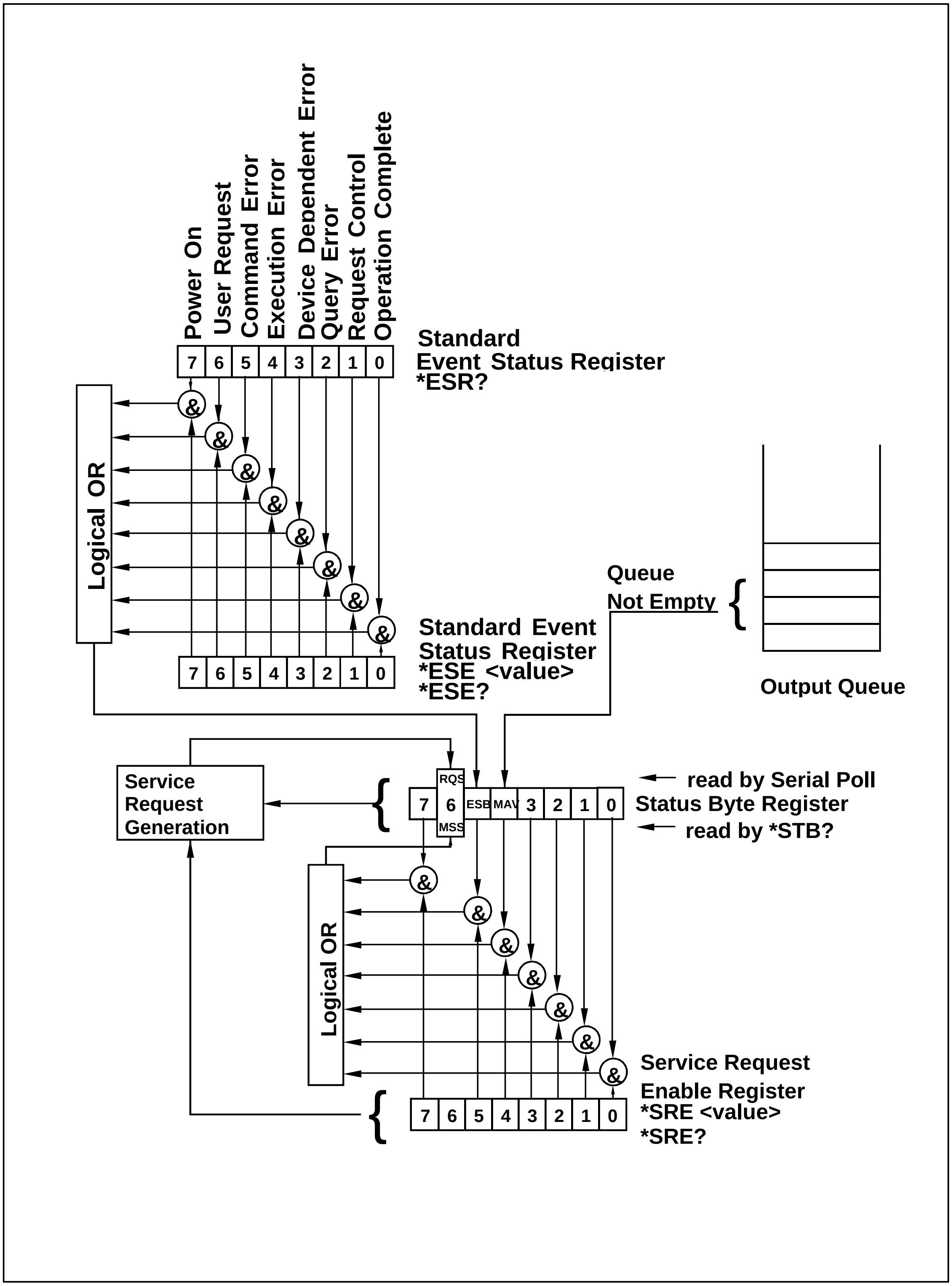

Figure 5-6, SCPI Status Registers

***SRE0** - Clears all bits in the register.
***SRE1** - Not used.
***SRE2** - Not used.
***SRE4** - Not used.
***SRE8** - Not used.
***SRE16** - Service request on MAV.
***SRE32** - Service request on ESB summary bit.
***SRE128** - Not used.

Standard Event Status Register (ESR)

The Standard Event Status Register reports status for special applications. The 8 bits of the ESR have been defined by the IEEE-STD-488.2 as specific conditions, which can be monitored and reported back to the user upon request. The Standard Event Status Register is destructively read with the *ESR? common query. The Standard Event Status Register is cleared with a *CLS common command, with a power-on and when read by *ESR?.

The arrangement of the various bits within the register is firm and is required by all GPIB instruments that implement the IEEE-STD-488.2. Description of the various bits is given in the following:

Bit 0 - Operation Complete. Generated in response to the *OPC command. It indicates that the device has completed all selected and pending operations and is ready for a new command.

Bit 1 - Request Control. This bit operation is disabled on the 3172.

Bit 2 - Query Error. This bit indicates that an attempt is being made to read data from the output queue when no output is either present or pending.

Bit 3 - Device Dependent Error. This bit is set when an error in a device function occurs. For example, the following command will cause a DDE error:

```
VOLTage 5;:VOLTage:OFFSet 2
```

Both of the above parameters are legal and within the specified limits, however, the generator is unable to generate such an amplitude and offset combination.

Bit 4 - Execution Error. This bit is generated if the parameter following the command is outside of the legal input range of the generator.

Bit 5 – Command Error. This bit indicates the generator received a command that was a syntax error or a command that the device does not implement.

Bit 6 - User Request. This event bit indicates that one of a set of local controls had been activated. This event bit occurs regardless of the remote or local state of the device.

Bit 7 - Power On. This bit indicates that the device's power source was cycled since the last time the register was read.

Standard Event Status Enable Register (ESE)

The Standard Event Status Enable Register allows one or more events in the Standard Event Status Register to be reflected in the ESB summary message bit. The Standard Event Status Enable Register is an 8-bit register that enables corresponding summary messages in the Standard Event Status Register. Thus, the application programmer can select reasons for the generator to issue an ESB summary message bit by altering the contents of the ESE Register.

The Standard Event Status Enable Register is read with the *ESE? Common query. The response to this query is a number that represents the sum of the binary-weighted value of the Standard Event Status Enable Register.

The Standard Event Status Enable Register is written using the *ESE command followed by a decimal value representing the bit values of the Register. A bit value one indicates an enabled condition. Consequently, a bit value of zero indicates a disabled condition. The Standard Event Status Enable Register is cleared by setting *ESE0. Summary of *ESE messages is given in the following.

***ESE0** – No mask. Clears all bits in the register.
***ESE1** – ESB on Operation Complete.
***ESE2** – ESB on Request Control.
***ESE4** – ESB on Query Error.
***ESE8** – ESB on Device Dependent Error.
***ESE16** – ESB on Execution Error.
***ESE32** – ESB on Command Error.
***ESE64** – ESB on User Request.
***ESE128** – ESB Power on.

Error Messages

In general, whenever the 3172 receives an invalid SCPI command, it automatically generates an error. Errors are stored in a special error queue and may be retrieved from this buffer one at a time. Errors are retrieved in first-in-first-out (FIFO) order. The first error returned is the first error that was stored. When you have read all errors from the queue, the generator responds with a 0,"No error" message.

If more than 30 errors have occurred, the last error stored in the queue is replaced with -350, “Queue Overflow”. No additional errors are stored until you remove errors from the queue. If no errors have occurred when you read the error queue, the generator responds with 0,"No error".

The error queue is cleared when power has been shut off or after a *CLS command has been executed. The *RST command does not clear the error queue. Use the following command to read the error queue:

```
SYSTem:ERRor?
```

Errors have the following format (the error string may contain up to 80 characters):

-102,"Syntax error"

A complete listing of the errors that can be detected by the generator is given below.

-100,"Command error". When the generator cannot detect more specific errors, this is the generic syntax error used.

-101,"Invalid Character". A syntactic element contains a character, which is invalid for that type.

-102,"Syntax error". Invalid syntax found in the command string.

-103,"Invalid separator". An invalid separator was found in the command string. A comma may have been used instead of a colon or a semicolon. In some cases where the generator cannot detect a specific separator, it may return error -100 instead of this error.

-104,"Data type error". The parser recognized a data element different than allowed.

-108,"Parameter not allowed". More parameters were received than expected for the header.

-109,"Missing parameter". Too few parameters were received for the command. One or more parameters that were required for the command were omitted.

-128."Numeric data not allowed". A legal numeric data element was received, but the instrument does not accept one in this position.

-131,"Invalid suffix". A suffix was incorrectly specified for a numeric parameter. The suffix may have been misspelled.

-148,"Character data not allowed". A character data element was encountered where prohibited by the instrument.

-200,"Execution error". This is the generic syntax error for the instrument when it cannot detect more specific errors. Execution error as defined in IEEE-488.2 has occurred.

-221,"Setting conflict". Two conflicting parameters were received which cannot be executed without generating an error. Listed below are events causing setting conflicts.

1. Sum of pulse or ramp parameters is more than 100 percent. Corrective action: Change parameters to correct the problem.
2. |ampl/2| + |offset| is more than 22 Vp-p (SYMM mode), 20 Vpk (POS or NEG mode). Corrective action: Reduce offset to 0, then change amplitude-offset values to correct the problem.
3. Activating filters when the 3172 is set to output the built-in sine waveform, or activating the built-in sine waveform when one of the 3172 filters is turned on. Corrective action: If in sine, select another function and activate the filter(s).
4. Activating burst mode when the 3172 is set to sequence mode, or activating sequence mode when the 3172 is set to burst mode. Corrective action: Remove the 3172 from burst or sequence and

then selected the desired mode.

5. Changing operating mode from triggered to continuous when the 3172 is set to single sequence advance, or changing the operating mode from continuous to triggered when the 3172 is set to automatic sequence advance mode. Corrective action: Observe the 3172 advance mode while setting sequence advance.

There are other settings conflict errors which are exclusively dedicated for the pulse function. These errors are listed and described in Chapter 3, under the pulse function description.

-222,"Data out of range". Parameter data, which followed a specific header, could not be used because its value is outside the valid range defined by the generator.

-224,"Illegal parameter value". A discrete parameter was received which was not a valid choice for the command. An invalid parameter choice may have been used.

-300,"Device-specific-error". This is the generic device-dependent error for the instrument when it cannot detect more specific errors. A device- specific error as defined in IEEE-488.2 has occurred.

-311,"Memory error". Indicates that an error was detected in the instrument's memory.

-350,"Queue Overflow". The error queue is full because more than 30 errors have occurred. No additional errors are stored until the errors from the queue are removed. The error queue is cleared when power has been shut off, or after a *CLS command has been executed.

-410,"Query INTERRUPTED". A command was received which sends data to the output buffer, but the output buffer contained data from a previous command (the previous data is not overwritten). The output buffer is cleared when power is shut off or after a device clear has been executed.

This page was intentionally left blank.

Chapter 6
Performance Checks

What's in this Chapter

This chapter provides performance tests necessary to verify proper operation of the Model 3172. The 3172 module can contain a combination of the following modules:

- W6: Single-channel arbitrary waveform generator
- P2: Dual-channel pulse generator
- A3: High voltage power amplifier

The available combinations are listed in Chapter 1. Perform only the performance checks for the modules installed in your 3172 unit.

WARNING

The procedures described in this section are for use only by qualified service personnel. Many of the steps covered in this section may expose the individual to potentially lethal voltages that could result in personal injury or death if normal safety precautions are not observed.

CAUTION

ALWAYS PERFORM PERFORMANCE TESTS AT A STATIC-SAFE WORKSTATION.

Performance Checks

The following performance checks verify proper operation of the instrument and should normally be used:

1. As a part of the incoming inspection of the instrument specifications;
2. As part of the troubleshooting procedure;
3. After any repair or adjustment before returning the instrument to regular service.

Environmental Conditions

Tests should be performed under laboratory conditions having an ambient temperature of 25° C, ±5° C and at relative humidity of less than 80%. If the instrument has been subjected to conditions outside these ranges, allow at least one additional hour for the instrument to stabilize before beginning the adjustment procedure. Specifications are valid within an ambient temperature of 25° C, ±5° C and at relative humidity of less than 80%. Below 20° C and above 30° C, the specifications are degraded by 0.1% for every ±1° C change

Warm-up Period

Most equipment is subject to a small amount of drift when it is first turned on. To ensure accuracy, turn on the power to the Model 3172-W6 and allow it to warm-up for at least 30 minutes before beginning the performance test procedure.

Initial Instrument Setting

To avoid confusion as to which initial setting is to be used for each test, it is required that the instrument be reset to factory default values prior to each test. To reset the Model 3172-W6 to factory defaults, use the Factory Rest option in the Utility Panel.

Recommended Test Equipment

Recommended test equipment specifications for troubleshooting, calibration and performance checking is listed in Table 6-1.

Table 6-1, Recommended Test Equipment Specifications

Equipment	Specification
Oscilloscope 500 MHz, 2 channels, jitter package	500 MHz, 2 channels, jitter package
Distortion Analyzer 1 MHz BW, Automatic distortion	1 MHz BW, Automatic distortion measurement < 0.02% distortion
measurement < 0.02% distortion	6.5 Digits, ACV, DCV
Digital Multimeter 6.5 Digits, ACV, DCV	200 MHz Universal counter timer, 2 channels, 1 ns single shot resolution
Spectrum Analyzer	1.5 GHz, +/- 0.4 dB accuracy
Pulse Generator (with manual trigger)	50 MHz, manual or software trigger

Test Procedures – W6 Module

The W6 module is a single-channel arbitrary waveform generator. Use the following procedures to check the W6 module against its specifications. A complete set of specifications is listed in Appendix A. The following paragraphs show how to set up the instrument for the test, what the specifications for the tested function are, and what acceptable limits for the test are. If the instrument fails to perform within the specified limits, the instrument must be calibrated or tested to find the source of the problem.

Initial Instrument Setting

To avoid confusion as to what initial setting is to be used for each test, it is required that instrument be reset to factory default values prior to each test.

Frequency Accuracy

Frequency accuracy checks tests the accuracy of the internal oscillator. The internal oscillator determines the accuracy and stability of the entire generator. The accuracy of the frequency depends on the 10 MHz reference oscillator. The W6 defaults to the internal TCXO that provides an accuracy of 1 ppm. The accuracy of the output frequency tests the internal TCXO because its accuracy is much higher than the backplane CLK10. If both the internal TCXO and the backplane CLK10 are insufficient for accuracy purposes, an external 10 MHz reference clock can be applied to the W6. The 10 MHz external reference input is not available for the legacy 3171 mode.

Frequency Accuracy, Internal Reference

Equipment: Counter

Preparation:

1. Configure the counter as follows:
   Termination: 50 Ω, DC coupled
2. Connect the W6 output to the counter input – channel A
3. Configure the W6 as follows:
   Waveform: Square wave
   10 MHz Source: Internal (TCXO)
   Amplitude: 2 V
   Output: On
   Frequency: As specified in Table 2

Test Procedure:

1. Perform frequency Accuracy tests using Table 6-2

Table 6-2, Frequency Accuracy

W6 Setting	Error Limits	Counter Reading	Pass	Fail
10.0000000 Hz	±10 µHz			
100.000000 Hz	±0.1 mHz			
1.00000000 kHz	±1 mHz			
10.0000000 kHz	±10 mHz			
100.000000 kHz	±100 mHz			
1.00000000 MHz	±1 Hz			
10.0000000 MHz	±10 Hz			
30.0000000 MHz	±30 Hz			

Frequency Accuracy, External 10 MHz Reference

Equipment: 10 MHz reference (at least 0.1ppm), Counter

Preparation:

1. Leave counter setting and W6 connections as in last test
2. Connect the 10 MHz reference oscillator to the W6 10 MHz reference input
3. Configure the W6 as follows:
   - 10 MHz: External
   - Waveform: Square wave
   - Amplitude: 2 V
   - Output: On
   - Frequency: As specified in Table 3

Test Procedure

1. Perform frequency Accuracy tests using Table 3

Table 6-3, Frequency Accuracy Using External 10 MHz Reference

W6 Setting	Error Limits	Counter Reading	Pass	Fail
10.000000000 MHz	±1 Hz			
20.000000000 MHz	±2 Hz			

Amplitude Accuracy

Amplitude accuracy checks tests the accuracy of the output amplifier and attenuators. The amplitude accuracy is checked for all three amplitude ranges: Symmetrical, Positive and Negative and for both the DAC route (arbitrary and standard waveforms) and the DDS route (CW and modulated waveforms).

Amplitude Accuracy, DAC Output

Equipment: DMM

Preparation:

1. Configure the DMM as follows:
   Termination: 50 Ω feedthrough at the DMM input
   Function: ACV
2. Connect W6 Channel to the DMM input
3. Configure the W6 as follows:
   Frequency: 1 kHz
   Output: On
   Amplitude Range: As specified in the test
   Amplitude: As specified in the

Test Procedure

1. Perform amplitude Accuracy tests using Tables 6-4 and 6-5.

Table 6-4, Amplitude Accuracy, DAC Output

W6 Amplitude Setting	Error Limits	DMM Reading (Each Range)			Pass	Fail
		Sym	Pos	Neg		
10.00 V	3.534 V, ±85.3 mV					
3.000 V	1.060 V, ±60.6 mV					
1.000 V	353.4 mV, ±23.5 mV					
200 mV	70.71 mV, ±20.7 mV					
100 mV	35.34 mV, ±5.3 mV					

2. Modify the amplitude range to Symmetrical and the output function to External AM. Perform amplitude accuracy checks using Table 6-5.

Table 6-5, Amplitude Accuracy – External AM, DAC Output

W6 Amplitude Setting	Error Limits	DMM Reading	Pass	Fail
10.00 V	3.534 V, ±85.3 mV			
3.000 V	1.060 V, ±60.6 mV			
1.000 V	353.4 mV, ±23.5 mV			
200 mV	70.71 mV, ±20.7 mV			
100 mV	35.34 mV, ±5.3 mV			

Amplitude Accuracy, DDS Output

Equipment: DMM

Preparation:

1. Configure the DMM as follows:
   Termination: 50 Ω feedthrough at the DMM input
   Function: ACV
2. Connect W6 output to the DMM input
3. Configure the W6 as follows:
   Waveform: Modulated
   Modulation: OFF
   CW Frequency: 1 kHz
   Output: On
   Amplitude Range: As specified in the test
   Amplitude: As specified in the test

Test Procedure

1. Perform amplitude Accuracy tests using Table 6-6.

Table 6-6, Amplitude Accuracy, DDS Output

W6 Amplitude Setting	Error Limits	DMM Reading (Each Range)			Pass	Fail
		Sym	Pos	Neg		
10.00 V	3.534 V, ±85.3 mV					
3.000 V	1.060 V, ±60.6 mV					
1.000 V	353.4 mV, ±23.5 mV					
200 mV	70.71 mV, ±20.7 mV					
100 mV	35.34 mV, ±5.3 mV					

Offset Accuracy

The offset accuracy is checked for all three amplitude ranges: Symmetrical, Positive and Negative and for both the DAC route (arbitrary and standard waveforms) and the DDS route (CW and modulated waveforms).

Offset Accuracy, DAC Output

Equipment: DMM

Preparation:

1. Configure the DMM as follows:
   Termination: 50 Ω feedthrough at the DMM input
   Function: DCV
2. Connect W6 output to the DMM input
3. Configure the W6 as follows:
   Frequency: 1 MHz
   Amplitude: 20 mV
   Output: On
   Amplitude Range: As specified in the test
   Offset: As specified in the test

Test Procedure

1. Perform Offset Accuracy tests using Table 6-7.

Table 6-7, Offset Accuracy – Symmetrical Range, DAC Output

W6 Offset Setting	Error Limits	DMM Reading	Pass	Fail
+4.000 V	4.000 V, ±55 mV			
+1.500 V	1.500 V, ±30 mV			
0.000 V	0 V, ±15 mV			
-1.500 V	-1.500 V, ±30 mV			
-4.000 V	-4.000 V, ±55 mV			

2. Modify W6 amplitude range to symmetrical, amplitude setting to 6 V and offset setting to 0 V.
3. Continue the Offset tests using Table 6-8.

Table 6-8, Offset Accuracy – Symmetrical Range, DAC Output - Continued

W6 Offset Setting	Error Limits	DMM Reading	Pass	Fail
0.000 V	0 ±65 mV			

4. Modify the amplitude range to Positive and perform offset accuracy checks using Table 6-9.

Table 6-9, Amplitude Accuracy – Positive Range, DAC output

W6 Offset Setting	Error Limits	DMM Reading	Pass	Fail
1.000 V	1.000 V, ±25 mV			
2.500 V	2.500 V, ±40 mV			
5.000 V	5.000 V, ±65 mV			

5. Modify the amplitude range to Negative and perform offset accuracy checks using Table 6-10.

Table 6-10, Amplitude Accuracy – Negative Range, DAC output

3172-W6 Offset Setting	Error Limits	DMM Reading	Pass	Fail
-1.000 V	-1.000 V, ±25 mV			
-2.500 V	-2.500 V, ±40 mV			
-5.000 V	-5.000 V, ±65 mV			

Offset Accuracy, DDS Output

Equipment: DMM

Preparation:

1. Configure the DMM as follows:
   Termination: 50 Ω feedthrough at the DMM input
   Function: DCV
2. Connect W6 output to the DMM input
3. Configure the W6 as follows:
   Waveform: Modulated
   Modulation: OFF
   CW Frequency: 1 MHz
   Amplitude: 6 V
   Output: On

Test Procedure

1. Perform Offset Accuracy tests using Table 6-11

Table 6-11, Offset Accuracy, DDS Output

W6 Offset Setting	Error Limits	DMM Reading	Pass	Fail
0.000 V	0 ±65 mV			

Square Wave Characteristics

This tests the characteristics of the square waveform. It includes transition times, ringing and overshoot. The characteristics are being tested on all three amplitude ranges: symmetrical, positive and negative.

Square Wave Checks

Equipment: Oscilloscope, 50 Ω, 20 dB attenuator feed through

Preparation:

1. Configure the Oscilloscope follows:
   Termination: 50 Ω, 20 dB attenuator feed through at the oscilloscope input
   Setup: As required for the test
2. Connect W6 output to the oscilloscope input
3. Configure the W6 as follows:
   Frequency: 1 MHz
   Waveform: Square wave
   Amplitude Range: As specified in the test
   Amplitude: 6 V
   Output: On

Test Procedure

1. Perform Squarewave Characteristics tests using Table 6-12.

Table 6-12, Square wave Characteristics

Parameter Tested	Error Limits	Oscilloscope Reading			Pass	Fail
		Sym	Pos	Neg		
Rise/Fall Time	<11 ns					
Ringing	<6.5 % + 10 mV					
Over/undershoot	<6.5 % + 10 mV					

2. Modify the amplitude setting to 10 V and continue with the square wave checks using Table 6-13.

Table 6-13, Square wave Characteristics, Continued

Parameter Tested	Error Limits	Oscilloscope Reading			Pass	Fail
		Sym	Pos	Neg		
Rise/Fall Time	<11 ns					
Ringing	<6.5 % + 10 mV					
Over/undershoot	<6.5 % + 10 mV					

Sinewave Characteristics

This tests the characteristics of the sine waveform. It includes distortion, spectral purity and flatness. Tests are done for both the DAC route (arbitrary and standard waveforms) and the DDS route (CW and modulated waveforms).

Sinewave Distortion, DAC Output

Equipment: Distortion Analyzer, Spectrum Analyzer, and ArbConnection

Preparation:

1. Connect W6 output to the distortion analyzer input. Configure the W6 as follows:
   SCLK: As required by the test
   Function: Arbitrary
   Amplitude Range: As specified in the test
   Amplitude: 5 V
   Output: On
2. Using ArbConnection prepare and download the following waveform:
   Waveform: Sinewave
   Wavelength: As required by the test

Test Procedure

1. Perform Sinewave distortion tests using Table 6-14.

Table 6-14, Sinewave Distortion, DAC Output Tests

W6 SCLK Settings	Sinewave Points	W6 Frequency	Reading Limits	Distortion Reading			Pass	Fail
				Sym	Pos	Neg		
400 kS/s	4000	100.0 Hz	< 0.2%					
4 MS/s	4000	1.000 kHz	< 0.2%					
40 Ms/s	4000	10.00 kHz	< 0.2%					
100 Ms/s	2000	50.00 kHz	< 0.2%					
100 Ms/s	1000	100.00 kHz	< 0.2%					

Sinewave Spectral Purity, DAC Output

Equipment: Spectrum Analyzer

Preparation:

1. Connect W6 output to the spectrum analyzer input. Use 50Ω and 20dB feedthrough termination at the spectrum analyzer input
2. Configure the W6 as follows:
   Amplitude Range: As specified in the test
   Amplitude: 5 V
   Output: On
   Frequency: As required by the test

Test Procedure

1. Perform sinewave spectral purity, DAC waveforms tests using Table 6-15

Table 6-15, Sinewave Spectral Purity, DAC Output Test

W6 Freq Settings	Reading Limits	Spectrum Analyzer		Reading			Pass	Fail
		Start	Stop	Sym	Pos	Neg		
1 MHz	>44 dBc	100 kHz	10 MHz					
5 MHz	>44 dBc	1 MHz	20 MHz					
10 MHz	>44 dBc	1 MHz	100 MHz					
30 MHz	>29 dBc	10 MHz	150 MHz					

2. Change amplitude to 10V and perform sine wave spectral purity, DAC waveforms tests using Table 6-16.

Table 6-16, Sinewave Spectral Purity, DAC Output Test – Continued

W6 Freq Settings	Reading Limits	Spectrum Analyzer		Reading			Pass	Fail
		Start	Stop	Sym	Pos	Neg		
1 MHz	>35 dBc	100 kHz	10 MHz					
5 MHz	>35 dBc	1 MHz	20 MHz					
10 MHz	>35 dBc	1 MHz	100 MHz					
30 MHz	>25 dBc	1 MHz	150 MHz					

Sinewave Spectral Purity, DDS Output

Equipment: Spectrum Analyzer

Preparation:

1. Connect W6 output to the spectrum analyzer input. Use 50 Ω and 20 dB feedthrough termination at the spectrum analyzer input
2. Configure the W6 as follows:
   Function: Modulated
   Modulation: OFF
   Amplitude Range: As specified in the test
   Amplitude: 5 V
   Output: On
   CW Frequency:As required by the test

Test Procedure

1. Perform sinewave spectral purity, DDS Waveforms tests using Table 6-17.

Table 6-17, Sine Wave Spectral Purity, DDS Output Tests

W6 Freq Settings	Reading Limits	Spectrum Analyzer		Reading			Pass	Fail
		Start	Stop	Sym	Pos	Neg		
1 MHz	>44 dBc	100 kHz	10 MHz					
5 MHz	>44 dBc	1 MHz	20 MHz					
10 MHz	>44 dBc	1 MHz	100 MHz					
30 MHz	>29 dBc	1 MHz	150 MHz					

2. Change amplitude to 10V and perform sine wave spectral purity, DDS waveforms tests using Table 6-18.

Table 6-18, Sine Wave Spectral Purity, DDS Output Tests – Continued

W6 Freq Settings	Reading Limits	Spectrum Analyzer		Reading			Pass	Fail
		Start	Stop	Sym	Pos	Neg		
1 MHz	>35 dBc	100 kHz	10 MHz					
5 MHz	>35 dBc	1 MHz	20 MHz					
10 MHz	>35 dBc	1 MHz	100 MHz					
30 MHz	>25 dBc	1 MHz	150 MHz					

Sinewave Flatness, DAC Output

Equipment: Oscilloscope

Preparation:

1. Configure the Oscilloscope follows:
   - Termination: 50 Ω, 20 dB feedthrough attenuator at the oscilloscope input
   - Setup: As required for the test
2. Connect W6 output to the oscilloscope input
3. Configure the W6 as follows:
   - Amplitude: 5 V
   - Output: On
   - Frequency: Initially, 1 kHz then, as required by the test

Test Procedure

1. Adjust the vertical controls of the Oscilloscope to get 6 division of display
2. Perform Sine flatness, DAC waveforms tests using Table 6-19.

Table 6-19, Sinewave Flatness, DAC Output Test

W6 Sine Frequency	Error Limits	Oscilloscope Reading	Pass	Fail
1 kHz	6 Divisions	Reference	X	X
1 MHz	6 ±0.6 Divisions			
10 MHz	6 ±0.6 Divisions			
20 MHz	6 ±1.8 Divisions			
30 MHz	6 ±1.8 Divisions			

3. Change amplitude to 10V and adjust the vertical controls of the Oscilloscope to get 6 division of display. Perform sine wave flatness, DAC waveforms tests using Table 6-20.

Table 6-20, Sinewave Flatness, DAC Output Test - Continued

W6 Sine Frequency	Error Limits	Oscilloscope Reading	Pass	Fail
1 kHz	6 Divisions	Reference	X	X
1 MHz	6 ±0.6 Divisions			
10 MHz	6 ±0.6 Divisions			
20 MHz	6 ±1.8 Divisions			
30 MHz	6 ±1.8 Divisions			

Sinewave Flatness, DDS Output

Equipment: Oscilloscope

Preparation:

1. Configure the Oscilloscope follows:
   - Termination: 50 Ω, 20 dB feedthrough attenuator at the oscilloscope input
   - Setup: As required for the test
2. Connect W6 output to the oscilloscope input
3. Configure the W6 as follows:
   - Function: Modulated
   - Modulation: OFF
   - Amplitude: 5 V
   - Output: On
   - CW Frequency: Initially, 1 kHz then, as required by the test

Test Procedure

1. Adjust the vertical controls of the Oscilloscope to get 6 division of display.
2. Perform Sine flatness, DDS waveforms tests using Table 6-21.

Table 6-21, Sinewave Flatness Test, DDS Output

W6 Sine Frequency	Error Limits	Oscilloscope Reading	Pass	Fail
1 kHz	6 Divisions	Reference	X	X
1 MHz	6 ±0.6 Divisions			
10 MHz	6 ±0.6 Divisions			
20 MHz	6 ±1.8 Divisions			
30 MHz	6 ±1.8 Divisions			

3. Change amplitude to 10V and adjust the vertical controls of the Oscilloscope to get 6 division of display. Perform sine wave flatness, DDS waveforms tests using Table 6-22.

Table 6-22, Sinewave Flatness Test, DDS Output - Continued

W6 Sine Frequency	Error Limits	Oscilloscope Reading	Pass	Fail
1 kHz	6 Divisions	Reference	X	X
1 MHz	6 ±0.6 Divisions			
10 MHz	6 ±0.6 Divisions			
20 MHz	6 ±1.8 Divisions			
30 MHz	6 ±1.8 Divisions			

Source Impedance Characteristics

This tests the accuracy of the source impedance. The W6 has three source impedances that can be used: <2 Ω, 50 Ω and 93 Ω. The usage of the source impedance depends on the characteristics of the load impedance. Regardless of the source impedance the W6 is of driving its full output swing into minimum of 50 Ω. Test the accuracy of the source impedance if you suspect that this is a problem with the output levels at different source impedance settings.

Source Impedance

Equipment: DMM

Preparation:

1. Configure the DMM follows:
   - Termination: 50 Ω feedthrough at the DMM input
   - Function: ACV
   - Range: 2 V
2. Connect W6 output to the DMM input
3. Configure the W6 as follows:
   - Frequency: 1 kHz
   - Amplitude: 5 V
   - Output: On

Test Procedure

1. Perform source impedance checks using Table 6-23.

Table 6-23, Source Impedance Accuracy Test

W6 Source Impedance	Low Limit	High Limit	DMM Reading	Pass	Fail
50 Ω	1.741	1.794			
<2 Ω	1.697	1.856			
93 Ω	1.723	1.829			

Trigger Operation Characteristics

This tests the operation of the trigger circuit. It includes tests for the triggered, gated and counted bursts run modes. It also tests the operation of the trigger advance options, the delayed trigger and re-trigger functions, as well as the trigger input level and slope sensitivity.

Trigger, Gate, and Burst Characteristics

Equipment: Oscilloscope, function generator, counter

Preparation:

1. Configure the Oscilloscope as follows:
   Termination: 50 Ω, 20d B feedthrough attenuator at the oscilloscope input
   Setup: As required for the test
2. Configure the counter as follows:
   Function: TOT B
   Trigger Level: 100 mV
3. Connect W6 output to the oscilloscope input
4. Configure the function generator as follows:
   Frequency 1 MHz
   Run Mode: As required by the test
   Wave: 2 V Square
5. Connect the function generator output to the W6 TRIG IN connector
6. Configure the W6 as follows:
   Frequency: 25 MHz
   Waveform: Sinewave
   Burst Count: 1e6 counts
   Amplitude: 1 V
   Trigger Source: External
   Output: On

Test Procedure

1. Perform trigger and gate tests using Table 6-24.

Table 6-24, Trigger, Gate, and Burst Characteristics

W6 Run Mode	External Trigger Pulse	Oscilloscope/Counter Reading	Pass	Fail
Triggered	1 MHz, Continuous	Triggered waveform		
Gated – Transition	1 MHz, Continuous	Gated by transition Waveform		
Gated – Level	1 MHz, Continuous	Gated by level Waveform		
Burst	Single shot	Burst, 1e6 waveforms		

Delayed Trigger Characteristics

Equipment: Function generator, 50 Ω “T” connector, Counter, ArbConnection CAD

Preparation:

1. Configure the Function generator as follows:
   - Amplitude: 1 V
   - Frequency: 1 MHz
   - Trigger Mode: Triggered.
   - Wave: Square Wave
2. Place the “T” connector on the output terminal of the function generator. Connect one side of the “T” to the W6 TRIG IN connector and the other side of the “T” to the channel A input of the counter
3. Connect the W6 output to channel B input of the counter
4. Configure the counter to TI A to B measurements
5. Using ArbConnection prepare and download the following waveform:
   - Wavelength: 100 points
   - Waveform: Pulse, Delay = 0.1, Rise/Fall = 0, High Time = 99.99
6. Configure the W6, channel 1 only, as follows:
   - SCLK: 100 MS/s
   - Waveform: Arbitrary
   - Run Mode: Triggered
   - Trigger Level 0 V
   - Trigger Delay: On
   - Delay: As required for the test
   - Amplitude: 5 V
   - Trigger Source: External
   - Output: On

Test Procedure

1. Perform trigger delay tests using Table 6-25.

Table 6-25, Trigger Delay Tests

3172-W6 Delay Setting	Error Limits	Counter Reading	Pass	Fail
1 μs	1 μs ±330 ns			
1 ms	1 ms ±50 μs			
1 s	1 s ±50 ms			
5 s	5 s ±500 ms			

Re-trigger Characteristics

Equipment: Counter, ArbConnection

Preparation:

1. Configure the counter as follows:
   Function: Pulse Width Measurement
   Ch A Slope: Negative
2. Connect the counter channel A to the W6 output
3. Using ArbConnection prepare and download the following waveform:
   Wavelength: 100 points
   Waveform: Pulse, Delay = 0.1, Rise/Fall = 0, High Time = 99.9
4. Configure the W6 as follows:
   SCLK 100 MS/s
   Waveform: Arbitrary
   Amplitude: 5 V
   Run Mode: Triggered
   Trigger Level 0 V
   Re-trigger: On
   Re-trigger Delay: As required by the test
   Trigger Source: Bus
   Output: On

Test Procedure

1. Manually trigger the instrument
2. Perform trigger delay tests using Table 6-26.

Table 6-26, Re-Trigger Delay Tests

W6 Re-trigger Setting	Error Limits	Counter Reading	Pass	Fail
1 μs	1 μs ±85 ns			
1 ms	1 ms ±50 μs			
1 s	1 s ±50 ms			
5 s	5 s ±500 ms			

Trigger Slope

Equipment: Oscilloscope, function generator

Preparation:

1. Configure the Oscilloscope follows:
   Termination: 50 Ω, 20 dB feedthrough attenuator at the oscilloscope input
   Setup: As required for the test
   Trigger Source: External
2. Connect W6 output to the oscilloscope input
3. Configure the function generator as follows:
   Frequency 10 kHz
   Run Mode: Continue
   Waveform: 2 V Square
4. Connect the function generator TTL output to the W6 TRIG IN connector
5. Connect the function generator main output to the 2nd channel of the oscilloscope
6. Configure the W6 as follows:
   Frequency: 1 MHz
   Waveform: Sine wave
   Run Mode: Triggered
   Output: On

Test Procedure

1. Toggle W6 trigger slope from positive to negative vice versa
2. Verify on the oscilloscope that the W6 transitions are synchronized with the slope of the trigger

Test Results	Pass		Fail	

Trigger Level

Equipment: Oscilloscope, function generator

Preparation:

1. Configure the Oscilloscope as follows:
   - Termination: 50 Ω, 20 dB feedthrough attenuator at the oscilloscope input
   - Setup: As required for the test
2. Connect W6 output to the oscilloscope input
3. Configure the function generator as follows:
   - Frequency 10 kHz
   - Run Mode: Continuous
   - Waveform: Squarewave.
   - Amplitude: 1 V
4. Connect the function generator output to the W6 TRIG IN connector
5. Configure the W6 as follows:
   - Frequency: 1 MHz
   - Waveform: Sine wave
   - Run Mode: Triggered
   - Trigger level: 0 V
   - Ch1 Output: On

Test Procedure

1. Verify that the W6 outputs triggered waveforms spaced at 0.1 ms
2. Modify the function generator offset to +2 V and change the W6 trigger level to +4 V. Verify that the W6 triggered waveforms are spaced 0.1 ms apart
3. Modify the function generator offset to -2 V and change the W6 trigger level to -4 V. Verify that the W6 triggered waveforms are spaced 0.1 ms apart

Test Results	Pass		Fail	

Backplane Trigger Source

Equipment: Oscilloscope, auxiliary W6 in an adjacent slot

Preparation:

1. Configure the Oscilloscope as follows:
   - Termination: 50 Ω, 20 dB feedthrough attenuator at the oscilloscope input
   - Setup: As required for the test
2. Connect the W6 output to the oscilloscope input
3. Configure the W6 as follows:
   - Frequency 1 MHz
   - Run Mode: Triggered
   - Run Mode Src: As specified in Table 6-26
   - Waveform: Sinewave
   - Amplitude: 2 V
   - Output: On
4. Configure the auxiliary W6 as follows:
   - Frequency: 2 MHz
   - Waveform: Sine wave
   - Run Mode: Continuous
   - Trigger Output: As specified in Table 6-27
   - Output: On

Test Procedure

1. Set up the trigger output and trigger source as specified in Table 6-27 and verify that the W6 generates a single-cycle, 2 MHz sinewave every 1 μs with every matched output trigger and source settings

Table 6-27, Trigger Source Tests

Auxiliary W6 Trigger Output Setting	W6 Trigger Source Setting	Oscilloscope Reading	Pass	Fail
TTLT0 OFF TTLT1 ON	TTLT1	1 μs trig intervals		
TTLT1 OFF TTLT2 ON	TTLT2	1 μs trig intervals		
TTLT2 OFF TTLT3 ON	TTLT3	1 μs trig intervals		
TTLT3 OFF TTLT4 ON	TTLT4	1 μs trig intervals		
TTLT4 OFF TTLT5 ON	TTLT5	1 μs trig intervals		
TTLT5 OFF TTLT6 ON	TTLT6	1 μs trig intervals		
TTLT6 OFF TTLT7 ON	TTLT7	1 μs trig intervals		

External SCLK Characteristics

This tests the operation of the arbitrary waveform function using an external sample clock source. Perform this test if you suspect that the accuracy of the output is degraded when using an external source, compared to when is used with an internal source.

External SCLK IN

Equipment: Counter, Function Generator

Preparation:

1. Configure the function generator as follows:
   Frequency: As required by the test
   Wave: Square
   Duty Cycle: 50%
   Amplitude 2 V
2. Connect the function generator to the W6 SCLK IN connector
3. Configure the W6, as follows:
   Mode: Arbitrary
   Amplitude: 5V
   Sample Clock: External
   Output: On
4. Using ArbConnection, create and download to the W6 a 100 points square waveform, single cycle

Test Procedure:

1. Change the function generator frequency and verify the output frequency as specified in Table 6-28.

Table 6-28, External Sample Clock Input Tests

Function Generator Frequency Setting	W6 Frequency	Pass	Fail
5 MHz	50 kHz, ±1 Hz		
50 MHz	500 kHz, ±10 Hz		

Sequence Operation

This tests the operation of the sequence generators. This also checks the various sequence advance options.

Automatic Advance

Equipment: Counter

Preparation:

1. Configure the Counter as follows:
   Function: TOTB Measurement
2. Connect the counter channel B to the W6 output
3. Configure the W6 as follows:
   SCLK 100 MS/s
   Waveform: Sequence
   Run Mode: Trigger
   Trigger Source: BUS
   Amplitude: 2 V
   Output: On
4. Using ArbConnection prepare and download the following waveform:
   Segments: 1 to 5
   Wavelength: 128 points
   Waveform: 1 cycle square
5. Using ArbConnection, build and download the following sequence table:
   Step 1: Segment 1, loop 100,000
   Step 2: Segment 2, loop 100,000
   Step 3: Segment 3, loop 100,000
   Step 4: Segment 4, loop 100,000
   Step 5: Segment 5, loop 100,000

Test Procedure

1. From ArbConnection, click on the Manual Trigger button and observe that counter reading is 500,000 counts. Reset counter and repeat the test a few times. Every time the counter reading should be 500,000 counts exactly

Test Results	Pass		Fail	

Step Advance

Equipment: Oscilloscope, function generator

Preparation:

1. Configure the Oscilloscope as follows:
   - Termination: 50 Ω, 20 dB feedthrough attenuator at the oscilloscope input
   - Setup: As required for the test
2. Connect the W6 output to the oscilloscope input
3. Configure the function generator as follows:
   - Frequency 10 kHz
   - Run Mode: Triggered
   - Waveform: Square Wave.
   - Amplitude: Adjust for TTL level on 50 Ω
4. Connect the function generator output to the W6 TRIG IN connector
5. Connect the W6 to the Oscilloscope input
6. Configure the W6 as follows:
   - SCLK 100 MS/s
   - Waveform: Sequence
   - Seq. Advance: Step
   - Amplitude: 2 V
   - Trigger Source: External
   - Output: On
7. Using ArbConnection prepare and download the following waveform:
   - Segment 1: Sine, 1000 points
   - Segment 2: Triangle, 1000 points
   - Segment 3: Square, 1000 points
   - Segment 4: Sinc, 1000 points
   - Segment 5: Gaussian Pulse, 1000 points
8. Using ArbConnection, build and download the following sequence table:
   - Step 1: Segment 1, loop 1
   - Step 2: Segment 2, loop 1
   - Step 3: Segment 3, loop 1
   - Step 4: Segment 4, loop 1
   - Step 5: Segment 5, loop 1

Test Procedure

1. Press the Manual Trigger on the function generator and observe that the waveforms advance through the sequence table repeatedly

Test Results	Pass		Fail	

Note

Leave the same setup for the next test

Single Advance

Equipment: Oscilloscope, function generator

Preparation: (Same preparation as for previous step, except change mode to single sequence advance)

1. Change Oscilloscope configuration to single

Test Procedure

1. Press the Manual Trigger on the function generator and observe that one cycle waveform advances through the sequence table repeatedly with each external trigger signal. Note that you need to press the Single mode on the oscilloscope for each trigger advance

Test Results	Pass		Fail	

SYNC Output Operation

This tests the operation of the SYNC output. There are two synchronous output are being tested – Bit and LCOM. Bit normally operates with standard and arbitrary waveforms and LCOM is always associated with sequenced and burst outputs. The sync output has fixed TTL level amplitude into an open circuit.

SYNC Output - Bit

Equipment: Oscilloscope

Preparation:

1. Configure the oscilloscope as follows:
   Time Base: As required by the test
   Amplitude: 2 V/div
2. Connect W6 SYNC output to the oscilloscope input
3. Configure model W6 as follows:
   Waveform: Sine
   Output: On
   Sync Output: On

Test Procedure:

1. Verify trace on the oscilloscope shows synchronization pulses at 1 μs intervals

Test Results	Pass		Fail	

SYNC Output - LCOM

Equipment: Oscilloscope

Preparation:

1. Configure the oscilloscope as follows:
   Time Base: As required by the test
   Amplitude: 2 V/div
2. Connect the W6 output to the oscilloscope input (1)
3. Connect the W6 SYNC output to the oscilloscope input (2)
4. Configure model W6 channel as follows:
   Waveform: Sine
   Run Mode: Burst
   Burst Count: 10
   Re-trigger On
   Re-trig period: 10 μs
   Trigger Source: BUS
   Output: On
   Sync Output: On

Test Procedure:

1. From ArbConnection, click on the Manual Trigger and verify that trace on the oscilloscope shows synchronization pulse having 10 μs pulse width. Verify that the SYNC is high for the duration of the burst.

Test Results	Pass		Fail	

SYNC Output - Pulse

Equipment: Oscilloscope

Preparation:

1. Configure the oscilloscope as follows:
   Time Base: As required by the test
   Amplitude: 2 V/div
2. Connect the W6 output to the oscilloscope input (1)
3. Connect the W6 SYNC output to the oscilloscope input (2)
4. Configure model W6 channel as follows:
   Waveform: Sine
   Output: On
   Sync Output: On
   Sync Source: Pulse
   Sync Width: 8
   Sync Position: 48

Test Procedure:

1. Observe that the SYNC output has 8 sample clock cycles width and that the position has shifted by 48 sample clock periods.

Test Results	Pass		Fail	

SYNC Output – Zero

Equipment: Oscilloscope

Preparation:

1. Configure the oscilloscope as follows:
   Time Base: As required by the test
   Amplitude: 2 V/div
2. Connect the W6 output to the oscilloscope input (1)
3. Connect the W6 SYNC output to the oscilloscope input (2)
4. Configure model 3172-W6 channel as follows:
   Waveform: Sinc
   ‘0’ Crossing 10
   Output: On
   Sync Output: On
   Sync Source: Zero

Test Procedure:

1. Observe that the SYNC output has 10 sinc cycles for each “0” crossing levels.

Test Results	Pass		Fail	

PLL Operation

This tests the operation of the PLL function. The W6 locks automatically to an external trigger source. The frequency and the start phase of the external signals are applied to the W6 settings. After lock, the start phase of the W6 can be modified from -180° to 180°.

PLL Checks – Frequency Lock

Equipment: Counter, function generator

Preparation:

1. Configure the function generator as follows:
   Waveform: Square
   Amplitude: 2 V
   Output: On
   Frequency: As required by the tests
2. Connect the function generator output to the W6 TRIG/PLL IN connector. Using a "T" connector, connect the same output to the counter input – CH A. Use 50 Ω feedthrough terminator at the W6 trigger input side
3. Configure the counter as follows:
   Function: Ratio A->B
   Input: 50 Ω
4. Connect the W6 output to the counter input – CH B.
5. Configure the model W6 as follows:
   Function: Arbitrary
   Waveform: Square, 20 points
   Output: On
   PLL: On

Test Procedure:

1. Modify the function generator frequency settings and observe that the counter readings match the function generator frequency settings, as specified in Table 6-29.

Table 6-29, PLL Tests – Frequency

Function Generator Frequency Setting	Error Limits	Counter Ratio Reading	Pass	Fail
500 Hz	1.00, ±0.01			
5 kHz	1.00, ±0.01			
50 kHz	1.00, ±0.01			
500 kHz	1.00, ±0.01			
5 MHz	1.00, ±0.01			
10 MHz	1.00, ±0.01			

PLL Checks – Phase Offset

Equipment: Counter, function generator

Preparation:

1. Configure the function generator as follows:
   Waveform: Square
   Amplitude: 2 V
   Output: On
   Frequency: As required by the tests
2. Connect the function generator output to the W6 TRIG/PLL IN connector. Using a “T” connector, connect the same output to the counter input – Channel A. Use 50 Ω feedthrough terminator at the W6 trigger input side
3. Configure the counter as follows:
   Function: ϕ A→B
   Input: 50 Ω
4. Connect the W6 output to the counter input – Channel B
5. Configure the model W6 as follows:
   Function Mode: Arbitrary
   Wavelength: 200 points
   Output: On
   PLL: On
   Phase Offset: As required by the test

Test Procedure:

1. Verify counter phase readings as specified in Table 6-30.

Table 6-30, PLL Tests – Phase Offset

Function Generator Frequency Setting	W6 Phase Setting	Error Limits	Counter Phase Reading	Pass	Fail
50 kHz	5°	5°, ±3°			
	90°	90°, ±3°			
	169°	169°, ±3°			
100 kHz	-5°	355°, ±3°			
	-90°	270°, ±3°			
	-169°	191°, ±3°			

PLL Checks – Backplane

Equipment: Oscilloscope, auxiliary W6 in an adjacent slot

Preparation:

1. Configure the oscilloscope as follows:
   - Termination: 50Ω, 20 dB feed through attenuator at the oscilloscope input
   - Setup: As required for the test
2. Connect the W6 output to the oscilloscope input
3. Configure the model W6 as follows:
   - Frequency 1 MHz
   - Run Mode: PLL
   - PLL Source: As specified in Table 31
   - Waveform: Sine wave
   - Amplitude: 2 V
   - Output: On
4. Configure the auxiliary W6 as follows:
   - Frequency: 10 kHz
   - Waveform: Sine wave
   - Run Mode: Continuous
   - Output: On
   - Trigger Output: As specified in Table 31

Test Procedure:

1. Verify backplane PLL lock using Table 6-31.

Table 6-31, PLL Tests – Phase Offset

Auxiliary W6 Backplane Trigger Setting	W6 PLL Source Setting	Oscilloscope Reading	Pass	Fail
TTLT0 ON	TTLT0			
ECLT1 ON	ECLT1			

PM Operation

This tests the operation of the PM function. After the W6 locks to an external trigger source, the start phase of the W6 can be modified in reference to the external signal using dc levels. The dc levels are applied to the PM input and control phase shifts of 20°/V.

PM Checks

Equipment: Counter, DC supply source, function generator

Preparation:

1. Configure the function generator as follows:
   Waveform: Square
   Amplitude: 2 V
   Output: On
   Frequency: 1 kHz
2. Using a “T” adapter, connect the function generator output to the W6 TRIG/PLL IN connector and the other cable to the counter input – Channel A. Use 50 Ω feedthrough terminator at the W6 trigger input side
3. Configure the counter as follows:
   Function: ϕ A→B
   Input: 50 Ω, both channels
4. Connect the W6 output to the counter input – Channel B
5. Configure the DC source as follows:
   Amplitude: As required by the test
6. Connect the DC source to the PM input
7. Configure the model W6 as follows:
   Function Mode: Arbitrary
   Waveform: Square
   Wavelength: 36 points
   Output: On
   PLL: On

Test Procedure:

1. Modify the DC source settings and verify readings as specified by the tests in Table 6-32.

Table 6-32, PLL Tests – PM Phase Offset

Freq Setting	DC Source Amplitude	Error Limits	Counter Phase Reading	Pass	Fail
10 kHz	2 V	320°,±15°			
10 kHz	4.5 V	270°,±15°			
10 kHz	-2 V	40°,±15°			
10 kHz	-4.5 V	90°,±15°			

Arbitrary Waveform Memory Operation

This tests the integrity of the waveform memory. The waveform memory stores the waveforms that are being generated at the output connector and therefore, flaws in the memory can cause distortions and impurity of the output waveforms.

Waveform Memory

Equipment: Distortion Analyzer, ArbConnection

Preparation:

1. Connect W6 output to the distortion analyzer input. Configure the W6 as follows:
   - SCLK: As required by the test
   - Waveform: Arbitrary
   - Amplitude: 5 V
   - Output: On
2. Using ArbConnection prepare and download the following waveform:
   - Wavelength: 1M points
   - Waveform: Sine wave
   - SCLK 100 MS/s

Test Procedure

1. Perform Sine wave distortion. It should be less than 0.1 %

Test Results	Pass		Fail	

Modulated Waveforms Operation

This tests the operation of the modulation circuits. It includes tests for the various modulation functions: FM, AM, FSK, PSK, Frequency hops and Sweep. Since the run modes are common to all modulation functions, they are being tested on the FM function only.

FM - Standard Waveforms

Equipment: Oscilloscope

Preparation:

1. Configure the oscilloscope as follows:
   Time Base: 50 µs
   Sampling Rate: 50 MS/s at least.
   Trace A View: Jitter, Type: FREQ, CLK.
   Trigger source: Channel 2, positive slope
   Amplitude: 1 V/div
2. Connect the W6 output to the oscilloscope input, channel 1
3. Connect the W6 SYNC output to the oscilloscope input, channel 2
4. Configure model W6 controls as follows:
   Waveform: Modulated
   Modulation: FM
   Carrier Freq: 1 MHz
   Mod Frequency: 10 kHz
   Deviation: 500 kHz
   Sync: On
   Output: On

Test Procedure:

1. Verify FM operation on the oscilloscope as follows:
   Waveform: Sine
   Frequency: 10 kHz
   Max A: 1.25 MHz
   Min A: 750 kHz

Test Results	Pass		Fail	

2. Modify W6 modulating waveform to triangle, then square and ramp and verify FM waveforms as selected

Test Results	Pass		Fail	

3. Move W6 marker position to 1.25MHz and verify marker position

Test Results	Pass		Fail	

Triggered FM - Standard Waveforms

Equipment: Oscilloscope, function generator

Preparation:

1. Configure the oscilloscope as follows:
   Time Base: 0.2 ms
   Sampling Rate: 50 MS/s at least.
   Trace A View: Jitter, Type: FREQ, CLK.
   Trigger source: Channel 2, positive slope
   Amplitude: 1 V/div
2. Connect the W6 output to the oscilloscope input, channel 1
3. Connect the W6 SYNC output to the oscilloscope input, channel 2
4. Configure the function generator as follows:
   Frequency 1 kHz
   Run Mode: Continuous
   Waveform: Squarewave.
   Amplitude: 2 V
5. Connect the function generator output connector to the W6 TRIG IN connector
6. Configure model W6 controls as follows:
   Waveform: Modulated
   Modulation: FM
   Mod Run Mode: Triggered
   Trigger Level: 0 V
   Carrier Freq: 1 MHz
   Mod Frequency: 10 kHz
   Deviation: 500 kHz
   Sync: On
   Output: On

Test Procedure:

1. Verify triggered FM – standard waveforms operation on the oscilloscope as follows:
   Waveform: Triggered sine waves
   Sine Frequency: 10 kHz
   Trigger Period: 1 ms
   Max A: 1.25 MHz
   Min A: 750 kHz

Test Results	Pass		Fail	

FM Burst - Standard Waveforms

Equipment: Oscilloscope, function generator

Preparation:

1. Configure the oscilloscope as follows:
   Time Base: 0.2 ms
   Sampling Rate: 50 MS/s at least.
   Trace A View: Jitter, Type: FREQ, CLK.
   Trigger source: Channel 2, positive slope
   Amplitude: 1 V/div
2. Connect W6 output to the oscilloscope input, channel 1
3. Connect the W6 SYNC output to the oscilloscope input, channel 2
4. Configure the function generator as follows:
   Frequency 1 kHz
   Run Mode: Continuous
   Waveform: Squarewave.
   Amplitude: 2 V Square
5. Connect the function generator output connector to the W6 TRIG IN connector
6. Configure model W6 controls as follows:
   Waveform: Modulated
   Modulation: FM
   Mod Run Mode: Burst
   Burst: 5
   Trigger Level: 0 V
   Carrier Freq: 1 MHz
   Mod Frequency: 10 kHz
   Deviation: 500 kHz
   Sync: On
   Output: On

Test Procedure:

1. Verify Burst FM – standard waveforms operation on the oscilloscope as follows:
   Waveform: Burst of 5 Sine waveforms
   Sine Frequency: 10 kHz
   Burst Period: 1 ms
   Max A: 1.25 MHz
   Min A: 750 kHz

Test Results	Pass		Fail	

Gated FM - Standard Waveforms

Equipment: Oscilloscope, function generator

Preparation:

1. Configure the oscilloscope as follows:
   Time Base: 0.2 ms
   Sampling Rate: 50 MS/s at least.
   Trace A View: Jitter, Type: FREQ, CLK.
   Trigger source: Channel 2, positive slope
   Amplitude: 1 V/div
2. Connect the W6 output to the oscilloscope input, channel 1
3. Connect the W6 SYNC output to the oscilloscope input, channel 2
4. Configure the function generator as follows:
   Frequency 1 kHz
   Run Mode: Continuous
   Waveform: Squarewave.
   Amplitude: 2 V
5. Connect the function generator output connector to the W6 TRIG IN connector
6. Configure model W6 controls as follows:
   Waveform: Modulated
   Modulation: FM
   Mod Run Mode: Gated
   Trigger Level: 0 V
   Carrier Freq: 1 MHz
   Mod Frequency: 10 kHz
   Deviation: 500 kHz
   Sync: On
   Output: On

Test Procedure:

1. Verify Gated FM – standard waveforms operation on the oscilloscope as follows:
   Waveform: Gated sine waveforms
   Sine Frequency: 10 kHz
   Gated Period: 1 ms
   Max A: 1.25 MHz
   Min A: 750 kHz

Test Results	Pass		Fail	

Re-triggered FM Bursts - Standard Waveforms

Equipment: Oscilloscope

Preparation:

1. Configure the oscilloscope as follows:
   Time Base: 0.2 ms
   Sampling Rate: 50 MS/s at least.
   Trace A View: Jitter, Type: FREQ, CLK.
   Trigger source: Channel 2, positive slope
   Amplitude: 1 V/div
2. Connect the W6 output to the oscilloscope input, channel 1
3. Connect the W6 SYNC to the oscilloscope input, channel 2
4. Configure model W6 controls as follows:
   Waveform: Modulated
   Modulation: FM
   Run Mode: Burst
   Burst Count: 5
   Carrier Freq: 1 MHz
   Mod Freq: 10 kHz
   Deviation: 500 kHz
   Sync: On
   Re-trigger: On
   Re-trig Delay: 200 μs
   Output: On

Test Procedure:

1. Verify re-triggered FM burst – standard waveforms operation on the oscilloscope as follows:
   Waveform: Repetitive burst of 5-cycle sine waveforms
   Sine Frequency: 10 kHz
   Re-trigger delay: 200 μs
   Max A: 1.25 MHz
   Min A: 750 kHz

Test Results	Pass		Fail	

FM - Arbitrary Waveforms

Equipment: Oscilloscope

Preparation:

1. Configure the oscilloscope as follows:
   Time Base: 0.2 ms
   Sampling Rate: 50 MS/s at least.
   Trace A View: Jitter, Type: FREQ, CLK.
   Trigger source: Channel 2, positive slope
   Amplitude: 1 V/div
2. Connect the W6 output to the oscilloscope input, channel 1
3. Connect the W6 SYNC to the oscilloscope input, channel 2
4. Configure model W6 controls as follows:
   Waveform: Modulated
   Modulation: FM
   Mod Waveform: Arbitrary
   Carrier Freq: 1 MHz
   FM SCLK: 2.5 MS/s
   Sync: On
   Output: On
5. Using ArbConnection prepare, open the FM Composer and download the following waveform:
   Wavelength: 4000 points
   Waveform: 4 cycles sinewave
   Deviation: 0.5 MHz

Test Procedure:

1. Verify FM operation on the oscilloscope as follows:
   Waveform: Sine
   Frequency: 2.5 kHz
   Max A: 1.25 MHz
   Min A: 750 kHz

Test Results	Pass		Fail	

AM

Equipment: Oscilloscope

Preparation:

1. Configure the oscilloscope as follows:
   Time Base: 20 μs
   Trigger source: Channel 2, positive slope
   Amplitude: 1 V/div
2. Connect the W6 output to the oscilloscope input, channel 1
3. Connect the W6 SYNC to the oscilloscope input, channel 2
4. Configure model W6 controls as follows:
   Waveform: Modulated
   Modulation: AM
   Carrier Freq: 1 MHz
   Mod Frequency: 1 kHz
   Mod Depth: 50 %
   Mod Wave Sine
   Sync: On
   Output: On

Test Procedure:

1. Verify AM operation on the oscilloscope as follows:
   Waveform: Amplitude modulated sine
   Mod depth: 50 % ±5 %

Test Results	Pass		Fail	

External AM

Equipment: Oscilloscope ,Function generator, “T” connector

Preparation:

1. Configure the oscilloscope as follows:
   Time Base: 0.2 ms
   Trigger source: Channel 2, positive slope
   Amplitude: 1 V/div
2. Configure the function generator as follows:
   Frequency: 1 KHz
   Trigger Mode: Continues
   Wave: Sinewave
3. Place the “T” connector on the output terminal of the function generator. Connect one side of the “T” to the W6 AM IN connector and the other side of the “T” to the channel 2 input of the oscilloscope
4. Connect the W6 output to the oscilloscope input, channel 1
5. Configure model W6 controls as follows:
   Function: External AM
   Output: On
   Carrier Freq: 1 MHz

Test Procedure:

1. Modify the amplitude of the external function generator and verify modulation depth operation on the oscilloscope using Table 6-33.

Table 6-33, Modulation Depth – External AM Tests

Function Gen Amplitude	Modulation Depth	Oscilloscope Reading	Pass	Fail
1 V	10 %			
5 V	50 %			
10 V	100 %			

FSK

Equipment: Oscilloscope

Preparation:

1. Configure the oscilloscope as follows:
   Time Base: 0.2 ms
   Sampling Rate: 50 MS/s at least.
   Trace A View: Jitter, Type: FREQ, CLK.
   Trigger source: Channel 2, positive slope
   Amplitude: 1 V/div.
2. Connect the W6 output to the oscilloscope input, channel 1
3. Connect the W6 SYNC to the oscilloscope input, channel 2
4. Configure model W6 controls as follows:
   Waveform: Modulated
   Modulation: FSK
   Carrier Freq: 2 MHz
   Shift Frequency: 4 MHz
   Baud Rate: 10 kHz
   Marker Index: 1
   Sync: On
   Output: On
5. Using ArbConnection, prepare and download 10-step FSK list with alternating “0” and “1”

Test Procedure:

1. Verify FSK operation on the oscilloscope as follows:
   Waveform: Squarewave
   Period: 0.2 ms
   Max Freq.: 4 MHz
   Min Freq.: 2 MHz

Test Results	Pass		Fail	

PSK

Equipment: Oscilloscope

Preparation:

1. Configure the oscilloscope as follows:
   Time Base: 0.2 ms
   Amplitude: 1 V/div.
2. Connect the W6 output to the oscilloscope input, channel 1
3. Connect the W6 SYNC to the oscilloscope input, channel 2
4. Configure model W6 controls as follows:
   Reset
   Waveform: Modulated
   Modulation: PSK
   Carrier Freq: 10 kHz
   Shift Phase: 180°
   Baud Rate: 10 kHz
   Sync: On
   Output: On
5. Using ArbConnection, prepare and download 10-step PSK list with alternating "0" and "1"

Test Procedure:

1. Verify PSK operation on the oscilloscope as follows:
   Waveform: Sinewave
   Period: 0.1 ms
   Phase: Every 0.1 ms change 180 degrees

Test Results	Pass		Fail	

ASK

Equipment: Oscilloscope

Preparation:

1. Configure the oscilloscope as follows:
   Time Base: 0.2 ms
   Amplitude: 1 V/div.
2. Connect the W6 output to the oscilloscope input, channel 1
3. Connect the W6 SYNC to the oscilloscope input, channel 2
4. Configure model W6 controls as follows:
   Reset
   Waveform: Modulated
   Modulation: ASK
   Carrier Freq: 10 kHz
   Base Amplitude: 4 V
   Shift Amplitude: 2 V
   Baud Rate: 10 kHz
   Sync: On
   Output: On
5. Using ArbConnection, prepare and download 10-step ASK list with alternating “0” and “1”

Test Procedure:

1. Verify ASK operation on the oscilloscope as follows:
   Waveform: Sinewave
   Period: 0.1 ms
   Amplitude: Every 0.1 ms alternates between 2 V and 4 V

Test Results	Pass		Fail	

Variable Dwell Time Frequency Hops

Equipment: Oscilloscope

Preparation:

1. Configure the oscilloscope as follows:
   Time Base: 0.5 ms
   Sampling Rate: 50 MS/s at least.
   Trace A View: Jitter, Type: FREQ, CLK.
   Trigger source: Channel 2, positive slope
   Amplitude: 1 V/div
2. Connect the W6 output to the oscilloscope input, channel 1
3. Connect the W6 SYNC to the oscilloscope input, channel 2
4. Configure model W6 controls as follows:
   Waveform: Modulated
   Modulation: Hop
   Hop Mode: Variable
   Sync: On
   Output: On
5. Using ArbConnection prepare, open the Hop Table composer and download the following table (both channels):

Frequency	*Dwell Time*
1.0e6	50e-6
1.2e6	100e-6
1.4e6	150e-6
1.6e6	200e-6
1.8e6	250e-6
2.0e6	300e-6
2.2e6	350e-6
2.4e6	400e-6
2.6e6	450e-6
2.8e6	500e-6

Test Procedure:

1. Verify Hop operation on the oscilloscope as follows:
   Waveform: Frequency steps, increasing dwell time from 50 μs to 500 μs
   Max A: 2.8 MHz
   Min A: 1.0 MHz
   Period: 2750 μs

Test Results	Pass		Fail	

Fix Dwell Time Frequency Hops

Equipment: Oscilloscope

Preparation:

1. Configure the oscilloscope as follows:
   Time Base: 0.1 ms
   Sampling Rate: 50 MS/s at least.
   Trace A View: Jitter, Type: FREQ, CLK.
   Trigger source: Channel 2, positive slope
   Amplitude: 1 V/div
2. Connect the W6 output to the oscilloscope input, channel 1
3. Connect the W6 SYNC to the oscilloscope input, channel 2
4. Configure model W6 controls as follows:
   Waveform: Modulated
   Modulation: Hop
   Hop Mode: Fix
   Dwell Time: 50 μs
   Sync: On
   Output: On
5. Using ArbConnection prepare, open the Hop Table composer and download the following table:

Frequency
1.0e6
1.2e6
1.4e6
1.6e6
1.8e6
2.0e6
2.2e6
2.4e6
2.6e6
2.8e6

Test Procedure:

1. Verify Hop operation on the oscilloscope as follows:
   Waveform: Frequency steps, fixed dwell time of 50 μs
   Max A: 2.8 MHz
   Min A: 1.0 MHz
   Period: 500 μs

Test Results	Pass		Fail	

Amplitude Hops

Equipment: Oscilloscope

Preparation:

1. Configure the oscilloscope as follows:
   Time Base: 50 μs
   Sampling Rate: 50 MS/s at least.
   Trigger source: Channel 2, positive slope
   Amplitude: 5 V/div
2. Connect the W6 output to the oscilloscope input, channel 1
3. Connect the W6 SYNC to the oscilloscope input, channel 2
4. Configure model W6 controls as follows:
   Waveform: Modulated
   Modulation: Amplitude Hops
   Hop Mode: Fix
   Dwell Time: 50 μs
   Sync: On
   Output: On
5. Using ArbConnection open and prepare the Hop Table composer and download the following table:

   Amplitude
   1
   2
   4
   8
   10

Test Procedure:

1. Verify 5 Hop operation on the oscilloscope as follows:
   Waveform: Amplitude steps, 50 μs fixed dwell time
   Min Amp: 1 V
   Max Amp: 10 V
   Period: 250 μs

Test Results	Pass		Fail	

Sweep

Equipment: Oscilloscope

Preparation:

1. Configure the oscilloscope as follows:
   Time Base: 0.2 ms
   Sampling Rate: 50 MS/s at least.
   Trace A View: Jitter, Type: FREQ, CLK.
   Trigger source: Channel 2, positive slope
   Amplitude: 1 V/div
2. Connect the W6 output to the oscilloscope input, channel 1
3. Connect the W6 SYNC output to the oscilloscope input, channel 2
4. Configure model W6 controls as follows:
   Waveform: Modulated
   Modulation: Sweep
   Start Frequency: 1 MHz
   Stop Frequency: 2 MHz
   Sweep Time: 1 ms
   Sweep Type: Linear
   Sync: On
   Output: On

Test Procedure:

1. Verify Sweep operation on the oscilloscope as follows:
   Waveform: Ramp up
   Frequency: 1 kHz
   Max A: 2 MHz
   Min A: 1 MHz

Test Results	Pass		Fail	

2. Move W6 sweep marker position to 1.5 MHz and verify marker position at the middle of the ramp

Test Results	Pass		Fail	

3. Reverse between Start and Stop frequencies and verify oscilloscope reading as before except the ramp is down

Test Results	Pass		Fail	

4. Change sweep step to logarithmic and verify oscilloscope exponential down waveform with properties as in 3 above

Test Results	Pass		Fail	

Auxiliary Counter/Timer Operation

This tests the operation of the auxiliary counter/timer function. Note that when you select the counter/timer function all other 3172-W6 waveform generation is automatically purged and the instrument operational mode is transformed to a stand-alone counter/timer. Waveform generation is resumed as soon as the counter/timer function is turned off.

Frequency

Equipment: Function Generator with at least 1 ppm accuracy

Preparation:

1. Configure the function generator as follows:
   Frequency: As required by the test
   Wave: Square
   Amplitude 500 mV
2. Connect the function generator to the W6 TRIG IN connector
3. Configure the W6, as follows:
   Auxiliary Function: Counter/Timer
   Function: Frequency
   Trigger Level: 0 V

Test Procedure:

1. Perform Frequency Measurement Accuracy tests using Table 6-34.

Table 6-34, Frequency Measurement Accuracy

Function Generator Setting	**Error Limits**	**3172-W6 Counter Reading**	**Pass**	**Fail**
1.000000 MHz	±2 Hz			
100.0000 MHz	±100 Hz			

Period, Period Averaged

Equipment: Function Generator with at least 1 ppm accuracy

Preparation:

1. Configure the function generator as follows:
   Frequency: As required by the test
   Wave: Square
   Amplitude 500 mV
2. Connect the function generator to the W6 TRIG IN connector
3. Configure the W6, as follows:
   Auxiliary Function: Counter/Timer
   Function: Period
   Trigger Level: 0 V

Test Procedure:

1. Perform Period Accuracy tests using Table 6-35.

Table 6-35, Period Measurement Accuracy

Function Generator Setting	Error Limits	3172-W6 Counter Reading	Pass	Fail
10 kHz	100.0 μs ±100 ns			
100 kHz	10.00 μs ±100 ns			

2. Change the counter/timer function to Period Averaged
3. With the last function generator setting in Table 6-35, verify that the period reading is 10.00000 μs ±50 ps

Test Results	Pass		Fail	

Pulse Width

Equipment: Function Generator with at least 1 ppm accuracy

Preparation:

1. Configure the function generator as follows:
   Frequency: As required by the test
   Wave: Square
   Duty Cycle: As required by the test
   Amplitude 500 mV
2. Connect the function generator to the W6 TRIG IN connector
3. Configure the W6, as follows:
   Auxiliary Function: Counter/Timer
   Function: Pulse Width
   Trigger Level: 0 V

Test Procedure:

1. Perform Pulse Width Accuracy tests using Table 6-36.

Table 6-36, Pulse Width Measurement Accuracy

Function Generator Setting		Error Limits	3172-W6 Counter Reading	Pass	Fail
Frequency	**Duty Cycle**				
100 kHz	50 %	5.000 µs ±100 ns			
100 kHz	70 %	7.00 µs ±100 ns			

2. Change the counter/timer slope to Negative
3. Verify that the pulse width reading is 3.0 µs ±100 ns

Test Results	Pass		Fail	

Digital Output

Equipment: Oscilloscope

Preparation:

1. Configure the oscilloscope as follows:
   Time Base: 0.2 ms
   Sampling Rate: 50 MS/s at least.
   Trace A View: Jitter, Type: FREQ, CLK.
   Trigger source: Channel 1, positive slope
   Amplitude: 1 V/div
2. Connect the W6 output to the oscilloscope input, channel 1
3. Configure model W6 controls as follows:
   Digital Output On

Test Procedure:

1. Verify Digital Signals on the Sync, Cursor and DIG0 to DIG9.

Test Results	Pass		Fail	

Test Procedures – P2 Module

The P2 module is a dual-channel pulse generator where each channel can be used separately or jointly, depending on the application on hand. Use the following procedures to check the P2 module against its specifications. A complete set of specifications is listed in Appendix A. The following paragraphs show how to set up the instrument for the test, what the specifications for the tested function are, and what are the acceptable limits for the specific test. If the instrument fails to perform within the specified limits, the instrument must be calibrated or tested to find the source of the problem.

P2 – Channel 1 Characteristics

The P2 module is comprised of two identical channels. Each is tested separately. The following tests check the performance and characteristics of the first channel only.

If channel 1 fails to perform in the specified limits, it must be recalibrated using the calibration procedure as outlined in Chapter 7 of this manual.

Initial Instrument Setting

To avoid confusion as to what initial setting is to be used for each test, it is required that instrument be reset to factory default values prior to each test.

Period Accuracy

This tests the accuracy of the pulse period. Pulse period is specified in two run modes: continuous and interrupted. In continuous mode, the accuracy is controlled by a Synthesizer circuit and hence the accuracy is much higher than in the interrupted run mode, where the accuracy there is controlled by an analog circuit. The accuracy is tested in two modes, continuous and gated only because the controlling circuit is the same for all of the interrupted run modes. To perform the tests without error conditions, reset the instrument and modify parameters that are specified in the tests only.

Period Accuracy, Continuous Run Mode Tests

Equipment: Counter/timer

Preparation:
1. Configure the counter/timer as follows:
   Function: Period Averaged
   Trigger Level: 0 V
   Termination: 50 Ω
2. Connect P2 channel 1 output to the counter/timer input. Configure the P2 as follows:
   Waveform: Single Pulse
   Output: On
   Pulse mode: Hold Duty Cycle
   Period: As required by the tests

Test Procedure
1. Perform period accuracy, continuous run mode tests using Table 6-37.

Table 6-37, Period Accuracy, Continuous Run Mode Tests

P2 Period Setting	Error Limits	Counter Reading	Pass	Fail
100.0000000 ms	±100 ns			
10.000000 ms	±10 ns			
1.000000 ms	±1 ns			
100.000 μs	±100 ps			
10.000 μs	±10 ps			
1.000 μs	±1 ps			

Period Accuracy, Gated Run Mode Tests

Equipment: Counter/timer, Pulse Generator

Preparation:
1. Configure the counter/timer as follows:
   Function: Period
   Trigger Level: 0 V
   Termination: 50 Ω
2. Connect P2 channel 1 output to the counter/timer input.
3. Configure the P2 as follows:
   Waveform: Single Pulse
   Run Mode: Gated
   Output: On
   Period: As required by the tests
   Pulse mode: Hold Duty Cycle

Test Procedure
1. Perform period accuracy, gated run mode tests using Table 6-38. Note that the reading will be stable during 2 seconds when the external pulse generator opens the gate. Discard other readings as irrelevant.

Table 6-38, Period Accuracy, Gated Run Mode Tests

P2 Period Setting	Error Limits	Counter Reading	Pass	Fail
100.0000000 ms	±3 ms			
10.000000 ms	±300 µs			
1.000000 ms	±30 µs			
100.000 µs	±3 µs			
10.000 µs	±300 ns			
1.000 µs	±30 ns			

Amplitude Accuracy

Amplitude accuracy checks tests the accuracy of the output amplifier and attenuators. The amplitude accuracy is checked for all three amplitude ranges: Symmetrical, Positive and Negative.

Amplitude Accuracy,

Equipment: DMM

Preparation:

1. Configure the DMM as follows:
   Termination: 50 Ω feedthrough at the DMM input
   Function: ACV
2. Connect P2 channel 1 to the DMM input.
3. Configure the P2 as follows:
   Frequency: 1 kHz
   Output: On
   Amplitude Range: As specified in the test
   Amplitude: As specified in the
   Pulse mode: Hold Duty Cycle

Test Procedure

3. Perform amplitude Accuracy tests using Table 6-39.

Table 6-39, Amplitude Accuracy, DAC output

P2 Amplitude Setting	Error Limits	DMM Reading (Each Range)			Pass	Fail
		Sym	Pos	Neg		
10.00 V	5.00 V, ±50 mV					
3.000 V	1.5 V, ±25 mV					
1.000 V	500 mV, ±18 mV					
200 mV	100 mV, ±15 mV					
100 mV	50 mV, ±15 mV					

Offset Accuracy

The offset accuracy is checked for all three amplitude ranges: Symmetrical, Positive and Negative. Test the accuracy of the offset if you suspect that this is a problem with the output amplifier.

Offset Accuracy,

Equipment: DMM

Preparation:

1. Configure the DMM as follows:
   Termination: 50 Ω feedthrough at the DMM input
   Function: DCV
2. Connect P2 channel 1 output to the DMM input
3. Configure the P2 as follows:
   Frequency: 1 MHz
   Amplitude: 20 mV
   Pulse mode: Hold Duty Cycle
   Output: On
   Amplitude Range: As specified in the test
   Offset: As specified in the test

Test Procedure

1. Perform Offset Accuracy tests using Table 40.

Table 6-40, Offset Accuracy – Symmetrical Range

P2 Offset Setting	Error Limits	DMM Reading	Pass	Fail
+4.000 V	4.000 V, ±35 mV			
+1.500 V	1.500 V, ±20 mV			
0.000 V	0 V, ±20 mV			
-1.500 V	-1.500 V, ±20 mV			
-4.000 V	-4.000 V, ±35 mV			

2. Modify the amplitude range to Positive and perform offset accuracy checks using Table 41.

Table 6-41, Amplitude Accuracy – Positive Range

P2 Offset Setting	Error Limits	DMM Reading	Pass	Fail
2.000 V	2.000 V, ±25 mV			
5.000 V	5.000 V, ±45 mV			
9.000 V	9.000 V, ±60 mV			

3. Modify the amplitude range to Negative and perform offset accuracy checks using Table 42.

Table 6-42, Amplitude Accuracy – Negative Range

P2 Offset Setting	Error Limits	DMM Reading	Pass	Fail
-2.000 V	-2.000 V, ±25 mV			
-5.000 V	-5.000 V, ±45 mV			
-9.000 V	-9.000 V, ±60 mV			

Source Impedance Characteristics

This tests the accuracy of the source impedance. The P2 has three source impedances that can be used: <2 Ω, 50 Ω and 93 Ω. The usage of the source impedance depends on the characteristics of the load impedance. Test the accuracy of the source impedance if you suspect that this is a problem with the output levels at different source impedance settings.

Source Impedance

Equipment: DMM

Preparation:

1. Configure the DMM follows:
   Termination: 50 Ω feedthrough at the DMM input
   Function: ACV
   Range: 20 V
2. Connect P2 channel 1 output to the DMM input
3. Configure the P2 as follows:
   Period: 1 ms
   Pulse mode: Hold Duty Cycle
   Amplitude: 5 V
   Output: On

Test Procedure

1. Perform source impedance checks using Table 43.

Table 6-43, Source Impedance Accuracy Test

P2 Source Impedance	Error Limits	DMM Reading	Pass	Fail
50 Ω	2.500, ±30 mV			
<2 Ω	2.500, ±100 mV			
93 Ω	1.748, ±100 mV			

Pulse Width Accuracy

This tests the accuracy of the pulse width. To eliminate counter threshold hysteresis problems the tests are performed with the fastest transitions only and at ranges that will not be effected by counter errors. To perform the tests without error conditions, reset the instrument and modify parameters that are specified in the tests only.

Pulse Width Accuracy Tests

Equipment: Counter/timer

Preparation:
1. Configure the counter/timer as follows:
   - Function: Pulse Width Averaged
   - Trigger Level: 0 V
   - Termination: 50 Ω
2. Connect P2 channel 1 output to the counter/timer input.
3. Configure the P2 as follows:
   - Waveform: Single Pulse
   - High Level: 2 V
   - Low Level: -2 V
   - Period: 100 ms
   - Output: On
   - Pulse Width: As required by the tests

Test Procedure
1. Perform pulse width accuracy tests using Table 6-44.

Table 6-44, Pulse Width Accuracy Tests

P2 Pulse Width Setting	Error Limits	Counter Reading	Pass	Fail
10.00000 ms	±10 µs			
1.000000 ms	±1.0 µs			
100.000 µs	±104 ns			
10.000 µs	±14 ns			
1.000 µs	±5 ns			
100 ns	±4.1 ns			

Pulse Delay, Double Pulse Delay Accuracy

This tests the accuracy of the pulse delay circuit. To eliminate counter threshold hysteresis problems the tests are performed with the fastest transitions only and at ranges that will not be effected by counter errors. For your information, the pulse delay and the double pulse delay share the same circuits. Also, the measurement of delayed pulse is more complicated because it involves manual subtraction of the SYNC to start delay and therefore, only double pulse delay is performed in this test and the results will verify the accuracy of the delayed pulse as well.

To perform the tests without error conditions, reset the instrument and modify parameters that are specified in the tests only.

Double Pulse Delay Accuracy Tests

Equipment: Counter/timer

Preparation:

1. Configure the counter/timer as follows:
   Function: Period
   Trigger Level: 0 V
   Termination: 50 Ω
2. Connect P2 channel 1 output to the counter/timer input.
3. Configure the P2 as follows:
   Waveform: Double Pulse
   Run Mode: Triggered
   High Level: 2 V
   Low Level: -2 V
   Period: 100 ms
   Pulse Width: 10 ns
   Output: On
   Dbl Pulse Delay: As required by the tests

Test Procedure

1. Manually trigger the P2 for each test.
2. Perform double pulse delay accuracy tests using Table 6-45. Reset counter reading after each test.

Table 6-45, Double Pulse Delay Accuracy Tests

P2 Double Pulse Delay Setting	Error Limits	Counter Reading	Pass	Fail
10.00000 ms	±10 μs			
1.000000 ms	±1.0 μs			
100.000 μs	±104ns			
10.000 μs	±14 ns			
1.000 μs	±5 ns			
100 ns	±4.1 ns			

Hold Duty Cycle Pulse Mode Accuracy

This tests the accuracy of the hold duty cycle pulse mode. Actually, the hold duty cycle mode is a special case of the single pulse mode except, in single pulse mode; the pulse width remains constant regardless of the period settings and in the hold duty cycle pulse mode, the ratio between the pulse width and the period remains constant regardless of the period settings. Note that each channel can have a unique duty cycle setting.

To perform the tests without error conditions, reset the instrument and modify parameters that are specified in the tests only.

Hold Duty Cycle Pulse Mode Accuracy Tests

Equipment: Counter/timer

Preparation:

1. Configure the counter/timer as follows:
   - Function: Pulse Width Averaged
   - Trigger Level: 0 V
   - Termination: 50 Ω
2. Connect P2 channels 1 output to the counter/timer input.
3. Configure the P2 as follows:
   - Waveform: Hold Duty Cycle
   - High Level: 2 V
   - Low Level: -2 V
   - Duty Cycle: 10%
   - Output: On
   - Period: As required by the tests

Test Procedure

1. Perform pulse width accuracy tests using Table 6-46.

Table 6-46, Hold Duty Cycle Pulse Mode Accuracy Tests

P2 Period Setting	Counter Period Reading Limits	Counter Reading	Pass	Fail
100.0000000 ms	10 ms ±300 µs			
10.000000 ms	1 ms ±30 µs			
1.000000 ms	100 µs ±3 µs			
100.000 µs	10 µs ±300.5 ns			
10.000 µs	1 µs ±30.5 ns			
1.000 µs	100 ns ±3.5 ns			

Linear Transitions Accuracy

This tests the accuracy of the transitions when the pulse is set to have linear transitions. Linear transitions imply that the slopes of the rise and fall times can be adjusted to have variable angles, other than the fastest upslope and down slope transitions. The transition times are measured from 10% to 90% of the amplitude setting, regardless of the high and low amplitude level settings. Linear transition control is independent for each channel however, one must keep in mind that the leading and trailing edges must remain within the same slope range boundaries and that the leading edge is the governing parameter, which means that the leading edge setting defines the transition range and the trailing edge must follow through.

To perform the tests without error conditions, reset the instrument and modify parameters that are specified in the tests only.

Linear Transitions Accuracy Tests

Equipment: Counter/timer

Preparation:
1. Configure the counter/timer as follows:
   Function: Rise Time Measurement function
   Termination: 50 Ω
2. Connect P2 channels 1 output to the counter/timer input.
3. Configure the P2 as follows:
   Waveform: Hold Duty Cycle
   Duty Cycle: 50%
   Transitions: Symmetrical
   Period: As required by the test
   High Level: 2 V
   Low Level: -2 V
   Output: On
   Leading Edge: As required by the tests

Test Procedure
1. Perform the leading edge linear transitions accuracy tests using Table 6-47.

Table 6-47, Leading Edge Transitions Accuracy Tests

P2 Leading Edge Setting	Period	Leading Edge Limits	Counter Reading	Pass	Fail
1.000 ms	10 ms	1 ms ±100 μs			
100.0 μs	1 ms	100 μs ±10 μs			
10.00 μs	100 μs	10 μs ±1 μs			
1.000 μs	10 μs	1 μs ±102 ns			
100 ns	1 μs	100 ns ±12 ns			
10 ns	100 ns	10 ns ±3 ns			

2. For the following tests modify the leading and trailing edge settings to be the identical. Modify the leading edge first.
3. Perform the trailing edge linear transitions accuracy tests using Table 6-48.

Table 6-48, Trailing Edge Transitions Accuracy Tests

P2 Trailing Edge Setting	Period	Trailing Edge Reading Limits	Counter Reading	Pass	Fail
1.000 ms	10 ms	1 ms ±100 μs			
100.0 μs	1 ms	100 μs ±10 μs			
10.00 μs	100 μs	10 μs ±1 μs			
1.000 μs	10 μs	1 μs ±102 ns			
100 ns	1 μs	100 ns ±12 ns			
10 ns	100 ns	10 ns ±3 ns			

External Pulse Width Mode Operation

This tests the operation of the external pulse width mode. This mode is particularly useful for reconstructing pulses from a week signal. Period and pulse width are derived from the trigger level and slope settings. The controlling signal is applied to the rear-panel TRIG IN connector. When the signal crosses the trigger threshold, it generates a pulse of which its width is determined by the inverse transition of the signal. Positive and negative slope settings determine if the width is derived from the positive trigger level crossing or the negative trigger level crossing.

To perform the tests without error conditions, reset the instrument and modify parameters that are specified in the tests only.

External Pulse Width Operation Tests

Equipment: Oscilloscope, Function Generator., 50Ω feedthrough termination

Preparation:

1. Configure the function generator as follows:
   Waveform: Square
   Level Output: TTL
   Frequency: 100 kHz
2. Connect the function generator to the P2 External Pulse Width input.
3. Connect P2 channel 1 output to the oscilloscope input
4. Configure the P2 as follows:
   Pulse Mode: Ext. Width
   Output: On

Test Procedure

1. Verify that the P2 generates pulses with the following properties:
   Period: 10 μs
   Pulse width: 5 μs

Test Results	Pass		Fail	

2. Change the P2 slope setting to negative and observe that the offset has a reverse impact on the pulse width.

Test Results	Pass		Fail	

Pulse Run Modes Operation

This tests the operation of the two specific pulse run modes: Internal Trigger and Internal Burst. Although all run modes characteristics are shared across the entire functionality of the P2, these two modes are specific for the pulse output. Operation of other run modes was tested under separate headings in this chapter.

To perform the tests without error conditions, reset the instrument and modify parameters that are specified in the tests only.

Pulse Run Modes Operation Tests

Equipment: Oscilloscope, Function Generator., 50 Ω feedthrough termination

Preparation:

1. Configure the function generator as follows:
   - Waveform: Square
   - Level Output: TTL
   - Frequency: 100 Hz
2. Connect the function generator to the P2 External Trigger input.
3. Connect P2 channel 1 output and Sync Output to the oscilloscope input.
4. Configure the P2 as follows:
   - Pulse Mode: Single
   - Pulse Width: 100 μs
   - Pulse Period: 200 μs
   - Output: On
   - Sync Output: On

Test Procedure

1. Verify trace on the oscilloscope showing synchronized pulses at 200 μs intervals (Pulse and Sync Outputs).
2. Change the P2 run mode setting to Trigger Verify that the P2 generates pulses with the following properties:
   - Period: 200 μs
   - Pulse width: 100 μs

   Also verify that the output and sync pulses are synchronized to the trigger signal at the Trigger Input.

Test Results	Pass		Fail	

3. Toggle P2 trigger slope from positive to negative and back.
4. Verify on the oscilloscope that the P2 transitions are synchronized with the slope of the trigger.

Test Results	Pass		Fail	

5. Change the P2 run mode setting to Burst and change the Burst Count setting to 5. Observe that 5 pulses are visible at every cycle of 1 ms.

Test Results	Pass		Fail	

6. Verify that Sync Output shows synchronization pulse having 5 ms pulse width. Verify that the SYNC is high for the duration of the burst.

Test Results	Pass		Fail	

7. Remove the P2 output from the oscilloscope input and connect to the counter input.
8. Change the counter function to Totalize.
9. Change the P2 burst count to 1,000 and internal timer to 1 second.
10. Reset counter and observe that the counter reading increments 1,000 counts every second.

Test Results	Pass		Fail	

11. Move function generator output to the P2 gate input.
12. Change the P2 run mode setting to Gated and observe that the pulse waveforms appear during the gate time only.

Test Results	Pass		Fail	

Delayed Trigger Characteristics

This tests the operation of the delayed trigger. Note that only one delay can be utilized at the time so, if you are using the double pulse mode, the delay is automatically associated with the double pulse delay and cannot be utilized for the trigger delay.

To perform the tests without error conditions, reset the instrument and modify parameters that are specified in the tests only.

Trigger Delay Tests

Equipment: Function generator, 50 Ω "T" connector, Counter, ArbConnection

Preparation:

1. Configure the function generator as follows:
   Amplitude: 1 V
   Frequency: 1 MHz
   Trigger Mode: Triggered.
   Wave: Square Wave
2. Place the "T" connector on the output terminal of the function generator. Connect one side of the "T" to the P2 TRIG IN connector and the other side of the "T" to the channel A input of the counter.
3. Connect the P2 channel 1 output to channel B input of the counter.
4. Configure the counter to TI A to B measurements.
5. Configure the P2, channel 1 only, as follows:
   Trigger Delay: On
   Delay: As required for the test
   Amplitude: 5 V
   Trigger Source: External
   Output: On

Test Procedure

1. Perform trigger delay tests using Table 6-49**.**

Table 6-49, Trailing Edge Transitions Accuracy Tests

P2 Delay Setting	Error Limits	Counter Reading	Pass	Fail
51 ns	51 ns ±2.5 ns			
105 ns	105 ns ±5 ns			
1 µs	1 µs ±230 ns			
1 ms	1 ms ±50 µs			
1 s	1 s ±50 ms			

Backplane Trigger Source

Equipment: Oscilloscope, auxiliary 3172 in an adjacent slot

Preparation:

1. Configure the Oscilloscope as follows:
   - Termination: 50 Ω, 20 dB feedthrough attenuator at the oscilloscope input
   - Setup: As required for the test
2. Connect the P2 output to the oscilloscope input
3. Configure the P2 as follows:
   - Frequency 1 MHz
   - Run Mode: Triggered
   - Run Mode Src: As specified in Table 6-50
   - Amplitude: 2 V
   - Output: On
4. Configure the auxiliary 3172 as follows:
   - Frequency: 100KHz
   - Waveform: Sine wave
   - Run Mode: Continuous
   - Trigger Output: As specified in Table 6-50
   - Output: On

Test Procedure

1. Set up the trigger output and trigger source as specified in Table 6-50 and verify that the P2 generates a pulses, every 10 μs with every matched output trigger and source settings.

Table 6-50, Trigger Source Tests

Auxiliary 3172 Trigger Output Setting	P2 Trigger Source Setting	Oscilloscope Reading	Pass	Fail
TTLT0 OFF TTLT1 ON	TTLT1	10 μs trig intervals		
TTLT1 OFF TTLT2 ON	TTLT2	10 μs trig intervals		
TTLT2 OFF TTLT3 ON	TTLT3	10 μs trig intervals		
TTLT3 OFF TTLT4 ON	TTLT4	10 μs trig intervals		
TTLT4 OFF TTLT5 ON	TTLT5	10 μs trig intervals		
TTLT5 OFF TTLT6 ON	TTLT6	10 μs trig intervals		
TTLT6 OFF TTLT7 ON	TTLT7	10 μs trig intervals		

Backplane Trigger - Output Source

Equipment: Oscilloscope, auxiliary 3172 in an adjacent slot

Preparation:

1. Configure the Oscilloscope as follows:
   Termination: 50Ω, 20 dB feed through attenuator at the oscilloscope input
   Setup: As required for the test
2. Configure the P2 as follows:
   Frequency: 1 MHz
   Output: On
   Trigger Source: TTL0
3. Configure the auxiliary 3172 as follows:
   Frequency: 2 MHz
   Waveform: Sine wave
   Run Mode: Triggered
   Trigger Input TTL0
   Output: On
4. Connect the auxiliary 3172 output to the oscilloscope input.

Test Procedure:

1. Set up the P2 trigger output source as specified in Table 6-50 and verify that the auxiliary 3172 generates a periodical single-cycle, 2 MHz sine with interval 1 µs.

Test Results	Pass		Fail	

Test Procedures – P2 Module

The P2 module is a dual-channel pulse generator where each channel can be used separately or jointly, depending on the application on hand. Use the following procedures to check the P2 module against its specifications. A complete set of specifications is listed in Appendix A. The following paragraphs show how to set up the instrument for the test, what the specifications for the tested function are, and what are the acceptable limits for the specific test. If the instrument fails to perform within the specified limits, the instrument must be calibrated or tested to find the source of the problem.

P2 – Channel 2 Characteristics

The P2 module is comprised of two identical channels. Each is tested separately. The following tests check the performance and characteristics of the second channel only.

If channel 2 fails to perform in the specified limits, it must be recalibrated using the calibration procedure as outlined in Chapter 7 of this manual.

Initial Instrument Setting

To avoid confusion as to what initial setting is to be used for each test, it is required that instrument be reset to factory default values prior to each test.

Period Accuracy

This tests the accuracy of the pulse period. Pulse period is specified in two run modes: continuous and interrupted. In continuous mode, the accuracy is controlled by a Synthesizer circuit and hence the accuracy is much higher than in the interrupted run mode, where the accuracy there is controlled by an analog circuit. The accuracy is tested in two modes, continuous and gated only because the controlling circuit is the same for all of the interrupted run modes. To perform the tests without error conditions, reset the instrument and modify parameters that are specified in the tests only.

Period Accuracy, Continuous Run Mode Tests

Equipment: Counter/timer

Preparation:
1. Configure the counter/timer as follows:
   Function: Period Averaged
   Trigger Level: 0 V
   Termination: 50 Ω
2. Connect P2 Channel 2 output to the counter/timer input. Configure the P2 as follows:
   Waveform: Single Pulse
   Output: On
   Pulse mode: Hold Duty Cycle
   Period: As required by the tests

Test Procedure
1. Perform period accuracy, continuous run mode tests using Table 6-51.

Table 6-51, Period Accuracy, Continuous Run Mode Tests

P2 Period Setting	Error Limits	Counter Reading	Pass	Fail
100.0000000 ms	±100 ns			
10.000000 ms	±10 ns			
1.000000 ms	±1 ns			
100.000 µs	±100 ps			
10.000 µs	±10 ps			
1.000 µs	±1 ps			

Period Accuracy, Gated Run Mode Tests

Equipment: Counter/timer, Pulse Generator

Preparation:
1. Configure the counter/timer as follows:
   Function: Period
   Trigger Level: 0 V
   Termination: 50 Ω
2. Connect P2 Channel 2 output to the counter/timer input.
3. Configure the P2 as follows:
   Waveform: Single Pulse
   Run Mode: Gated
   Output: On
   Period: As required by the tests
   Pulse mode: Hold Duty Cycle

Test Procedure
1. Perform period accuracy, gated run mode tests using Table 6-52. Note that the reading will be stable during 2 seconds when the external pulse generator opens the gate. Discard other readings as irrelevant.

Table 6-52, Period Accuracy, Gated Run Mode Tests

P2 Period Setting	Error Limits	Counter Reading	Pass	Fail
100.0000000 ms	±3 ms			
10.000000 ms	±300 µs			
1.000000 ms	±30 µs			
100.000 µs	±3 µs			
10.000 µs	±300 ns			
1.000 µs	±30 ns			

Amplitude Accuracy

Amplitude accuracy checks tests the accuracy of the output amplifier and attenuators. The amplitude accuracy is checked for all three amplitude ranges: Symmetrical, Positive and Negative.

Amplitude Accuracy,

Equipment: DMM

Preparation:

1. Configure the DMM as follows:
   - Termination: 50 Ω feedthrough at the DMM input
   - Function: ACV
2. Connect P2 Channel 2 to the DMM input
3. Configure the P2 as follows:
   - Frequency: 1 kHz
   - Output: On
   - Amplitude Range: As specified in the test
   - Amplitude: As specified in the
   - Pulse mode: Hold Duty Cycle

Test Procedure

1. Perform amplitude accuracy tests using Table 6-53.

Table 6-53, Amplitude Accuracy, DAC output

P2 Amplitude Setting	Error Limits	DMM Reading (Each Range)			Pass	Fail
		Sym	Pos	Neg		
10.00 V	5.00 V, ±50 mV					
3.000 V	1.5 V, ±25 mV					
1.000 V	500 mV, ±18 mV					
200 mV	100 mV, ±15 mV					
100 mV	50 mV, ±15 mV					

Offset Accuracy

The offset accuracy is checked for all three amplitude ranges: Symmetrical, Positive and Negative. Test the accuracy of the offset if you suspect that this is a problem with the output amplifier.

Offset Accuracy,

Equipment: DMM

Preparation:

1. Configure the DMM as follows:
   Termination: 50 Ω feedthrough at the DMM input
   Function: DCV
2. Connect P2 channel 2 output to the DMM input.
3. Configure the P2 as follows:
   Frequency: 1 MHz
   Amplitude: 20 mV
   Pulse mode: Hold Duty Cycle
   Output: On
   Amplitude Range: As specified in the test
   Offset: As specified in the test

Test Procedure

1. Perform Offset Accuracy tests using Table 54.

Table 6-54, Offset Accuracy – Symmetrical Range

P2 Offset Setting	Error Limits	DMM Reading	Pass	Fail
+4.000 V	4.000 V, ±35 mV			
+1.500 V	1.500 V, ±20 mV			
0.000 V	0 V, ±20 mV			
-1.500 V	-1.500 V, ±20 mV			
-4.000 V	-4.000 V, ±35 mV			

2. Modify the amplitude range to Positive and perform offset accuracy checks using Table 55.

Table 6-55, Amplitude Accuracy – Positive Range

P2 Offset Setting	Error Limits	DMM Reading	Pass	Fail
2.000 V	2.000 V, ±25 mV			
5.000 V	5.000 V, ±45 mV			
9.000 V	9.000 V, ±60 mV			

3. Modify the amplitude range to Negative and perform offset accuracy checks using Table 56.

Table 6-56, Amplitude Accuracy – Negative Range

P2 Offset Setting	Error Limits	DMM Reading	Pass	Fail
-2.000 V	-2.000 V, ±25 mV			
-5.000 V	-5.000 V, ±45 mV			
-9.000 V	-9.000 V, ±60 mV			

Source Impedance Characteristics

This tests the accuracy of the source impedance. The P2 has three source impedances that can be used: <2 Ω, 50 Ω and 93 Ω. The usage of the source impedance depends on the characteristics of the load impedance. Test the accuracy of the source impedance if you suspect that this is a problem with the output levels at different source impedance settings.

Source Impedance

Equipment: DMM

Preparation:

1. Configure the DMM follows:
   Termination: 50 Ω feedthrough at the DMM input
   Function: ACV
   Range: 20 V
2. Connect P2 channel 2 output to the DMM input
3. Configure the P2 as follows:
   Period: 1 ms
   Pulse mode: Hold Duty Cycle
   Amplitude: 5 V
   Output: On

Test Procedure

1. Perform source impedance checks using Table 57.

Table 6-57, Source Impedance Accuracy Test

P2 Source Impedance	Error Limits	DMM Reading	Pass	Fail
50 Ω	2.500, ±30 mV			
<2 Ω	2.500, ±100 mV			
93 Ω	1.748, ±100 mV			

Pulse Width Accuracy

This tests the accuracy of the pulse width. To eliminate counter threshold hysteresis problems the tests are performed with the fastest transitions only and at ranges that will not be effected by counter errors. To perform the tests without error conditions, reset the instrument and modify parameters that are specified in the tests only.

Pulse Width Accuracy Tests

Equipment: Counter/timer

Preparation:

1. Configure the counter/timer as follows:
   Function: Pulse Width Averaged
   Trigger Level: 0 V
   Termination: 50 Ω
2. Connect P2 channel 2 output to the counter/timer input.
3. Configure the P2 as follows:
   Waveform: Single Pulse
   High Level: 2 V
   Low Level: -2 V
   Period: 100 ms
   Output: On
   Pulse Width: As required by the tests

Test Procedure

1. Perform pulse width accuracy tests using Table 6-58.

Table 6-58, Pulse Width Accuracy Tests

P2 Pulse Width Setting	Error Limits	Counter Reading	Pass	Fail
10.00000 ms	±10 μs			
1.000000 ms	±1.0 μs			
100.000 μs	±104 ns			
10.000 μs	±14 ns			
1.000 μs	±5 ns			
100 ns	±4.1 ns			

Pulse Delay, Double Pulse Delay Accuracy

This tests the accuracy of the pulse delay circuit. To eliminate counter threshold hysteresis problems the tests are performed with the fastest transitions only and at ranges that will not be effected by counter errors. For your information, the pulse delay and the double pulse delay share the same circuits. Also, the measurement of delayed pulse is more complicated because it involves manual subtraction of the SYNC to start delay and therefore, only double pulse delay is performed in this test and the results will verify the accuracy of the delayed pulse as well.

To perform the tests without error conditions, reset the instrument and modify parameters that are specified in the tests only.

Double Pulse Delay Accuracy Tests

Equipment: Counter/timer

Preparation:

1. Configure the counter/timer as follows:
   Function: Period
   Trigger Level: 0 V
   Termination: 50 Ω
2. Connect P2 channel 2 output to the counter/timer input.
3. Configure the P2 as follows:
   Waveform: Double Pulse
   Run Mode: Triggered
   High Level: 2 V
   Low Level: -2 V
   Period: 100 ms
   Pulse Width: 10 ns
   Output: On
   Dbl Pulse Delay: As required by the tests

Test Procedure

1. Manually trigger the P2 for each test.
2. Perform double pulse delay accuracy tests using Table 6-59. Reset counter reading after each test.

Table 6-59, Double Pulse Delay Accuracy Tests

P2 Double Pulse Delay Setting	Error Limits	Counter Reading	Pass	Fail
10.00000 ms	±10 µs			
1.000000 ms	±1.0 µs			
100.000 µs	±104 ns			
10.000 µs	±14 ns			
1.000 µs	±5 ns			
100 ns	±4.1 ns			

Hold Duty Cycle Pulse Mode Accuracy

This tests the accuracy of the hold duty cycle pulse mode. Actually, the hold duty cycle mode is a special case of the single pulse mode except, in single pulse mode; the pulse width remains constant regardless of the period settings and in the hold duty cycle pulse mode, the ratio between the pulse width and the period remains constant regardless of the period settings. Note that each channel can have a unique duty cycle setting.

To perform the tests without error conditions, reset the instrument and modify parameters that are specified in the tests only.

Hold Duty Cycle Pulse Mode Accuracy Tests

Equipment: Counter/timer

Preparation:

1. Configure the counter/timer as follows:
   - Function: Pulse Width Averaged
   - Trigger Level: 0 V
   - Termination: 50 Ω
2. Connect P2 channel 2 output to the counter/timer input.
3. Configure the P2 as follows:
   - Waveform: Hold Duty Cycle
   - High Level: 2 V
   - Low Level: -2 V
   - Duty Cycle: 10%
   - Output: On
   - Period: As required by the tests

Test Procedure

1. Perform pulse width accuracy tests using Table 6-60.

Table 6-60, Hold Duty Cycle Pulse Mode Accuracy Tests

P2 Period Setting	Counter Period Reading Limits	Counter Reading	Pass	Fail
100.0000000 ms	10 ms ±300 µs			
10.000000 ms	1 ms ±30 µs			
1.000000 ms	100 µs ±3 µs			
100.000 µs	10 µs ±300.5 ns			
10.000 µs	1 µs ±30.5 ns			
1.000 µs	100 ns ±3.5 ns			

Linear Transitions Accuracy

This tests the accuracy of the transitions when the pulse is set to have linear transitions. Linear transitions imply that the slopes of the rise and fall times can be adjusted to have variable angles, other than the fastest upslope and down slope transitions. The transition times are measured from 10% to 90% of the amplitude setting, regardless of the high and low amplitude level settings. Linear transition control is independent for each channel however, one must keep in mind that the leading and trailing edges must remain within the same slope range boundaries and that the leading edge is the governing parameter, which means that the leading edge setting defines the transition range and the trailing edge must follow through.

To perform the tests without error conditions, reset the instrument and modify parameters that are specified in the tests only.

Linear Transitions Accuracy Tests

Equipment: Counter/timer

Preparation:
1. Configure the counter/timer as follows:
   - Function: Rise Time Measurement function
   - Termination: 50 Ω
2. Connect P2 channel 2 output to the counter/timer input.
3. Configure the P2 as follows:
   - Waveform: Hold Duty Cycle
   - Duty Cycle: 50%
   - Transitions: Symmetrical
   - Period: As required by the test
   - High Level: 2 V
   - Low Level: -2 V
   - Output: On
   - Leading Edge: As required by the tests

Test Procedure
1. Perform the leading edge linear transitions accuracy tests using Table 6-61.

Table 6-61, Leading Edge Transitions Accuracy Tests

P2 Leading Edge Setting	Period	Leading Edge Limits	Counter Reading	Pass	Fail
1.000 ms	10 ms	1 ms ±100 µs			
100.0 µs	1 ms	100 µs ±10 µs			
10.00 µs	100 µs	10 µs ±1 µs			
1.000 µs	10 µs	1 µs ±102 ns			
100 ns	1 µs	100 ns ±12 ns			
10 ns	100 ns	10 ns ±3 ns			

2. For the following tests modify the leading and trailing edge settings to be the identical. Modify the leading edge first.
3. Perform the trailing edge linear transitions accuracy tests using Table 6-62.

Table 6-62, Trailing Edge Transitions Accuracy Tests

P2 Trailing Edge Setting	Period	Trailing Edge Reading Limits	Counter Reading	Pass	Fail
1.000 ms	10 ms	1 ms ±100 µs			
100.0 µs	1 ms	100 µs ±10 µs			
10.00 µs	100 µs	10 µs ±1 µs			
1.000 µs	10 µs	1 µs ±102 ns			
100 ns	1 µs	100 ns ±12 ns			
10 ns	100 ns	10 ns ±3 ns			

External Pulse Width Mode Operation

This tests the operation of the external pulse width mode. This mode is particularly useful for reconstructing pulses from a week signal. Period and pulse width are derived from the trigger level and slope settings. The controlling signal is applied to the rear-panel TRIG IN connector. When the signal crosses the trigger threshold, it generates a pulse of which its width is determined by the inverse transition of the signal. Positive and negative slope settings determine if the width is derived from the positive trigger level crossing or the negative trigger level crossing.

To perform the tests without error conditions, reset the instrument and modify parameters that are specified in the tests only.

External Pulse Width Operation Tests

Equipment: Oscilloscope, Function Generator., 50Ω feedthrough termination

Preparation:

1. Configure the function generator as follows:
   Waveform: Square
   Level Output: TTL
   Frequency: 100 kHz
2. Connect the function generator to the P2 External Pulse Width input.
3. Connect P2 channel 2 output to the oscilloscope input.
4. Configure the P2 as follows:
   Pulse Mode: Ext. Width
   Output: On

Test Procedure

1. Verify that the P2 generates pulses with the following properties:
   Period: 10 μs
   Pulse width: 5 μs

Test Results	Pass		Fail	

2. Change the P2 slope setting to negative and observe that the offset has a reverse impact on the pulse width.

Test Results	Pass		Fail	

Pulse Run Modes Operation

This tests the operation of the two specific pulse run modes: Internal Trigger and Internal Burst. Although all run modes characteristics are shared across the entire functionality of the P2, these two modes are specific for the pulse output. Operation of other run modes was tested under separate headings in this chapter.

To perform the tests without error conditions, reset the instrument and modify parameters that are specified in the tests only.

Pulse Run Modes Operation Tests

Equipment: Oscilloscope, Function Generator., 50 Ω feedthrough termination

Preparation:

1. Configure the function generator as follows:
   Waveform: Square
   Level Output: TTL
   Frequency: 100 Hz
2. Connect the function generator to the P2 External Trigger input.
3. Connect P2 channel 2 outputs and Sync Output to the oscilloscope input.
4. Configure the P2 as follows:
   Pulse Mode: Single
   Pulse Width: 100 μs
   Pulse Period: 200 μs
   Output: On
   Sync Output: On

Test Procedure

1. Verify trace on the oscilloscope showing synchronized pulses at 200 μs intervals (Pulse and Sync Outputs).
2. Change the P2 run mode setting to Trigger. Verify that the P2 generates pulses with the following properties:
   Period: 200 μs
   Pulse width: 100 μs
   Also verify that the output and sync pulses are synchronized to the trigger signal at the Trigger Input**.**

Test Results	Pass		Fail	

3. Toggle P2 trigger slope from positive to negative and back.
4. Verify on the oscilloscope that the P2 transitions are synchronized with the slope of the trigger.

Test Results	Pass		Fail	

5. Change the P2 run mode setting to Burst and change the Burst Count setting to 5. Observe that 5 pulses are visible at every cycle of 1 ms.

Test Results	Pass		Fail	

6. Verify that Sync Output shows synchronization pulse having 5 ms pulse width. Verify that the SYNC is high for the duration of the burst.

Test Results	Pass		Fail	

7. Remove the P2 output from the oscilloscope input and connect to the counter input.
8. Change the counter function to Totalize.
9. Change the P2 burst count to 1,000 and internal timer to 1 second.
10. Reset counter and observe that the counter reading increments 1,000 counts every second.

Test Results	Pass		Fail	

11. Move function generator output to the P2 gate input.
12. Change the P2 run mode setting to Gated and observe that the pulse waveforms appear during the gate time only.

Test Results	Pass		Fail	

Delayed Trigger Characteristics

This tests the operation of the delayed trigger. Note that only one delay can be utilized at the time so, if you are using the double pulse mode, the delay is automatically associated with the double pulse delay and cannot be utilized for the trigger delay.

To perform the tests without error conditions, reset the instrument and modify parameters that are specified in the tests only.

Trigger Delay Tests

Equipment: Function generator, 50 Ω “T” connector, Counter, ArbConnection

Preparation:

1. Configure the function generator as follows:
   Amplitude: 1 V
   Frequency: 1 MHz
   Trigger Mode: Triggered.
   Wave: Square Wave
2. Place the “T” connector on the output terminal of the function generator. Connect one side of the “T” to the P2 TRIG IN connector and the other side of the “T” to the channel A input of the counter.
3. Connect the P2 channel 2 output to channel B input of the counter.
4. Configure the counter to TI A to B measurements.
5. Configure the P2, channel 2 only, as follows:
   Trigger Delay: On
   Delay: As required for the test
   Amplitude: 5 V
   Trigger Source: External
   Output: On

Test Procedure

1. Perform trigger delay tests using Table 6-63**.**

Table 6-63, Trailing Edge Transitions Accuracy Tests

P2 Delay Setting	Error Limits	Counter Reading	Pass	Fail
51 ns	51 ns ±2.5 ns			
105 ns	105 ns ±5 ns			
1 µs	1 µs ±230 ns			
1 ms	1 ms ±50 µs			
1 s	1 s ±50 ms			

Backplane Trigger Source

Equipment: Oscilloscope, auxiliary 3172 in an adjacent slot

Preparation:

1. Configure the Oscilloscope as follows:
   Termination: 50 Ω, 20 dB feedthrough attenuator at the oscilloscope input
   Setup: As required for the test
2. Connect the P2 output to the oscilloscope input
3. Configure the P2 as follows:
   Frequency 1 MHz
   Run Mode: Triggered
   Run Mode Src: As specified in Table 6-64
   Amplitude: 2 V
   Output: On
4. Configure the auxiliary 3172 as follows:
   Frequency: 100 KHz
   Waveform: Sine wave
   Run Mode: Continuous
   Trigger Output: As specified in Table 6-64
   Output: On

Test Procedure

1. Set up the trigger output and trigger source as specified in Table 6-64 and verify that the P2 generates a pulse every 10 μs with every matched output trigger and source settings.

Table 6-64, Trigger Source Tests

Auxiliary 3172 Trigger Output Setting	P2 Trigger Source Setting	Oscilloscope Reading	Pass	Fail
TTLT0 OFF TTLT1 ON	TTLT1	10 μs trig intervals		
TTLT1 OFF TTLT2 ON	TTLT2	10 μs trig intervals		
TTLT2 OFF TTLT3 ON	TTLT3	10 μs trig intervals		
TTLT3 OFF TTLT4 ON	TTLT4	10 μs trig intervals		
TTLT4 OFF TTLT5 ON	TTLT5	10 μs trig intervals		
TTLT5 OFF TTLT6 ON	TTLT6	10 μs trig intervals		
TTLT6 OFF TTLT7 ON	TTLT7	10 μs trig intervals		

Backplane Trigger - Output Source

Equipment: Oscilloscope, auxiliary 3172 in an adjacent slot

Preparation:

1. Configure the Oscilloscope as follows:
   - Termination: 50 Ω, 20 dB feed through attenuator at the oscilloscope input
   - Setup: As required for the test
2. Configure the P2 as follows:
   - Frequency: 1 MHz
   - Output: On
   - Trigger Source: TTL0
3. Configure the auxiliary 3172 as follows:
   - Frequency: 2 MHz
   - Waveform: Sine wave
   - Run Mode: Triggered
   - Trigger Input TTL0
   - Output: On
4. Connect the auxiliary 3172 output to the oscilloscope input.

Test Procedure:

1. Set up the P2 trigger output source as specified in Table 6-64 and verify that the auxiliary 3172 generates a periodical single-cycle, 2 MHz sine with interval 1 µs.

Test Results	Pass		Fail	

Two-Channel Synchronization

This tests the synchronization between the two P2 channels. The synchronization process assures that the two channels transition simultaneously and on the same edge with as little skew as possible between the first transitions. Perform the two-channel synchronization tests if you suspect that the two channels do not synchronize as expected.

To perform the tests without error conditions, reset the instrument and modify parameters that are specified in the tests only.

Synchronization Tests

Equipment: Oscilloscope, Counter

Preparation:

1. Connect the P2 channel 1 and channel 2 outputs to two oscilloscope inputs.
2. Configure channel 1 as follows:
   Waveform: Single
   Period: 15 μs
   Pulse width: 1 μs
   Output: On
3. Configure channel 2 as follows:
   Waveform: Single
   Period: 100 μs
   Pulse width: 1 μs
   Output: On
   Clock Source: Channel 1

Test Procedure

1. Verify on the oscilloscope that the period of channel 2 is 15 μs and is synchronized with the output of channel 1.

Test Results	Pass		Fail	

2. Modify the clock source as follows:
   Channel 1: Channel 2
   Channel 2: Channel 2
3. Verify on the oscilloscope that the period of channel 1 is 100 μs and is synchronized with the output of channel 2.

Test Results	Pass		Fail	

Test Procedures – A3 Module

The A3 module is a high voltage power amplifier that installs in the 3172 waveform generator module. The A3 operates in conjunction with the W6 arbitrary waveform generator module. Use the following procedures to check the A3 module against its specifications that are listed in Appendix A. The following paragraphs show how to set up the instrument for the test, the specifications for the tested function, and the acceptable test limits. If the instrument fails to perform within the specified limits, the instrument must be calibrated or tested to find the source of the problem.

Initial Instrument Setting

To avoid confusion as to what initial setting is to be used for each test, it is required that instrument be reset to factory default values prior to each test.

Gain Accuracy Test

Equipment: DMM

Preparation:

1. Configure the DMM as follows:
   Function: ACV
2. Connect the 3172-W6 output to DMM input.
3. Configure the 3172-W6 as follows:
   Wave Shape: SIN
   Frequency: 10 kHz
   Amplitude: As required for the test
   Output: On
4. Verifying DMM reading of 3172-W6 amplitude.
5. Connect the 3172-W6 output to the A3 input.
6. Connect the A3 output to the DMM input.

Test Procedure

1. Perform Gain Accuracy tests using Table 6-65.

Table 6-65, Gain Accuracy Tests

AMPLITUDE SETTING	ERROR LIMIT	DMM READING	PASS	FAIL
2(x12/Gain) V	8.48V, ±100mV			
4(x12/Gain) V	16.97V, ±200mV			
8(x12/Gain) V	33.94V, ±400mV			
10(x12/Gain) V	42.42V, ±500mV			

Bandwidth Test

Equipment: Oscilloscope, Load Resistance, x100 high voltage probe

Preparation:

1. Configure the Oscilloscope as follows:
   Time Base: 200 μs/div
   Amplitude: 50 V/div
2. Connect the A3 output to the load resistance.
3. Connect the x100 high voltage probe across the load resistance.
4. Configure the 3172-W6 as follows:
   Function: Standard
   Waveform: Sine
   Frequency: As required for the tests
   Amplitude: 10 Vpp (x12/Gain)
   Output: On
5. Connect the 3172-W6 output to the A3 Input.

Test Procedure:

1. Using the variable vertical adjustment on the oscilloscope, adjust the vertical trace to show exactly 6 vertical divisions.
2. Perform bandwidth, large signals tests using Table 6-66.

Table 6-66, Output Bandwidth, Large Signals Tests

FREQUENCY SETTING	ERROR LIMIT	OSCILLOSCOPE READING	PASS	FAIL
1 kHz	6 Divisions			
25 kHz	6 ±0.5 Divisions			
50 kHz	6 ±1 Divisions			
100 kHz	6 ±1.8 Divisions			

Rise/Fall time Tests

Equipment: Oscilloscope, Load Resistance, x100 high voltage probe

Preparation:

1. Configure the Oscilloscope as follows:
   Time Base: 200 μs/div
   Amplitude: 50 V/div
2. Connect the A3 output to the load resistance.
3. Connect the x100 high voltage probe across the load resistance.
4. Configure the 3172-W6 as follows:
   Function: Standard
   Waveform: Square
   Frequency: 10kHz
   Amplitude: 8Vpp
   Output: On
5. Connect the 3172-W6 output to the A3 Input.

Test Procedure:

1. Using the variable vertical adjustment on the oscilloscope, adjust the vertical trace to show exactly 6 vertical divisions.
2. Perform rise/fall time tests output using Table 6-67.

Table 6-67, Rise/Fall Time Tests

PARAMETER TESTED	ERROR LIMIT	OSCILLOSCOPE READING	PASS	FAIL
Rise Time	<1.5 μs			
Fall Time	<1.5 μs			

Overshoot Tests

Equipment: Oscilloscope, Load Resistance, x100 high voltage probe

Preparation:

1. Configure the Oscilloscope as follows:
   Time Base: 500 ns/div
   Amplitude: 50 V/div
2. Connect the A3 output to the load resistance.
3. Connect the x100 high voltage probe across the load resistance.
4. Configure the 3172-W6 as follows:
   Function: Standard
   Waveform: Square
   Frequency: 10 kHz
   Amplitude: 8 Vpp
   Output: On
5. Connect the 3172-W6 output to the A3 Input.

Test Procedure:

1. Using the variable vertical adjustment on the oscilloscope, adjust the vertical trace to show exactly 6 vertical divisions.
2. Perform overshoot tests using Table 6-68.

Table 6-68, Overshoot Tests

PARAMETER TESTED	ERROR LIMIT	OSCILLOSCOPE READING	PASS	FAIL
Overshoot	15%			

This page was intentionally left blank.

Chapter 7
Calibration and Firmware Update

What's in this Chapter

This chapter gives instructions for performing calibration and firmware updates of the 3172 VXIbus Arbitrary Waveform Generator.

WARNING

The procedures described in this section are for use only by qualified service personnel. Many of the steps covered in this section may expose the individual to potentially lethal voltages that could result in personal injury or death if normal safety precautions are not observed.

CAUTION

Always perform disassembly, repair, and cleaning at a static-safe work station.

Scope

The calibration sections in this chapter address the W6 and W2 waveform generator modules and the P2 pulse generator module. These designations refer to the module types indicated in the 3172 model number.

For example:

Model 3172-W6P2 contains one W6-type waveform generator module and one P2-type pulse generator module.

Model 3172-W6W6 contains two W6-type waveform generators.

Several other 3172 models exist, with various combinations of internal modules.

Calibration

Calibration Types

There are two types of calibration for Model 3172:

- **Factory Calibration:** Performed only on new instruments as part of the manufacturing process or after repair at an authorized service center.
- **Periodic Calibration:** Performed at recommended intervals to ensure that the instrument remains accurate over time.

Instructions for both calibration types are provided in this chapter.

Recommended Test Equipment

Recommended equipment for calibration is listed in Table 7-1. Other test instruments may be used only if their specifications equal or exceed the required characteristics.

Table 69, Recommended Test Equipment

Equipment	Model No.	Manufacturer
Digital Oscilloscope (with jitter package)*	LC684D	LeCroy
Digital Multi-meter (DMM)	2000	Keithley
Frequency Counter (Rubidium reference)*	2202R	Astronics Test Systems
Function Generator (with manual trigger)*	WW2571A	Tabor Electronics
Accessories	Coaxial cable: One end fitted with BNC male Other end fitted with contact part number DM53740 (Cinch or Cannon), or Astronics part number 602300-900	
	BNC “T” connector (2 BNC females with 1 BNC male)*	
	Adapter, dual banana jacks with BNC male	
	Feed-through termination, 50Ω ± 0.1%, 5W	

* Required for Factory Calibration only.

Environment

Although Model 3172 can operate at ambient temperatures from 0°C to 50°C, calibration should be performed under laboratory conditions with an ambient temperature of 25°C ± 5°C and relative humidity of less than 80%.

Configuration

Some calibration steps require removal of the right side panel (cover). To keep the internal temperature stable, leave the cover on during warm-up and calibration, except while making adjustments that require its removal. To facilitate removal while the unit is in the VXI chassis, remove all except two screws before installing the unit. Replace all screws after calibration is complete.

Calibration is performed with the 3172 installed in a VXIbus chassis. Either a register-based or message-based controller may be used, and the procedure is the same for both options. The ArbConnection software application must be installed and interfaced to the instrument.

Periodic Calibration

Calibration Interval

To ensure continued accuracy, it is recommended that periodic calibration be performed at intervals of one year.

Warm-up

Install the 3172 in a VXIbus chassis. Turn on the chassis power and allow the 3172 to warm up and stabilize for at least 30 minutes before calibrating. If the instrument has been subjected to temperatures outside the range of 25°C ± 5°C , allow at least one additional hour for the instrument to stabilize before calibrating.

Calibration Setup and Initialization

1. Start the ArbConnection software. The Startup & Communication Options window will appear as shown in Figure 7-1.

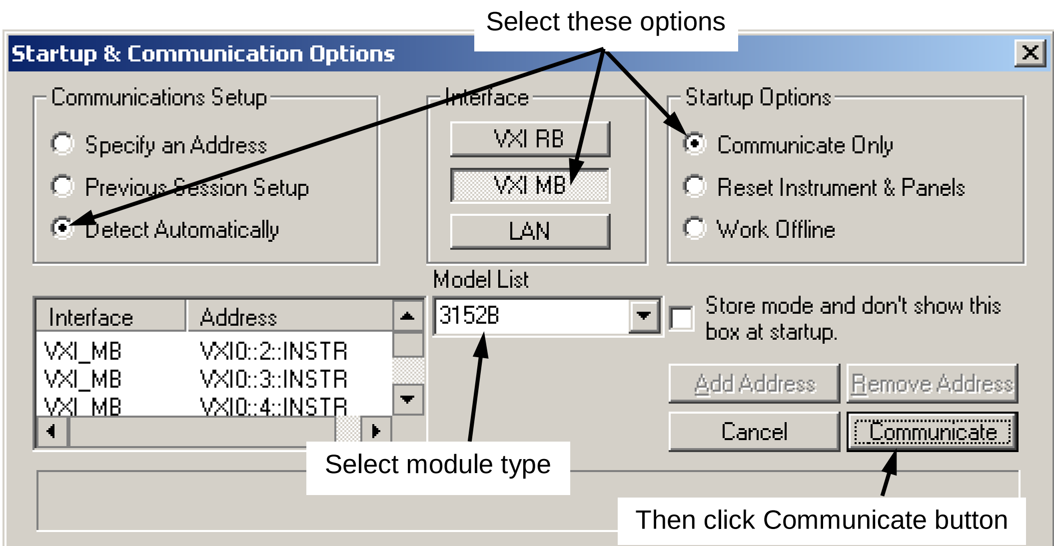

Figure 7-1, Startup & Communication Options Window

2. Select "Detect Automatically", "VXI MB", and "Communicate Only" as shown in Figure 7-1. Then click the "Communicate" button.
3. In the Model List box, select the model number, such as 3172-W6P2 or 3172-P2.
4. The ArbConnection main window will then appear (Figure 7-2).

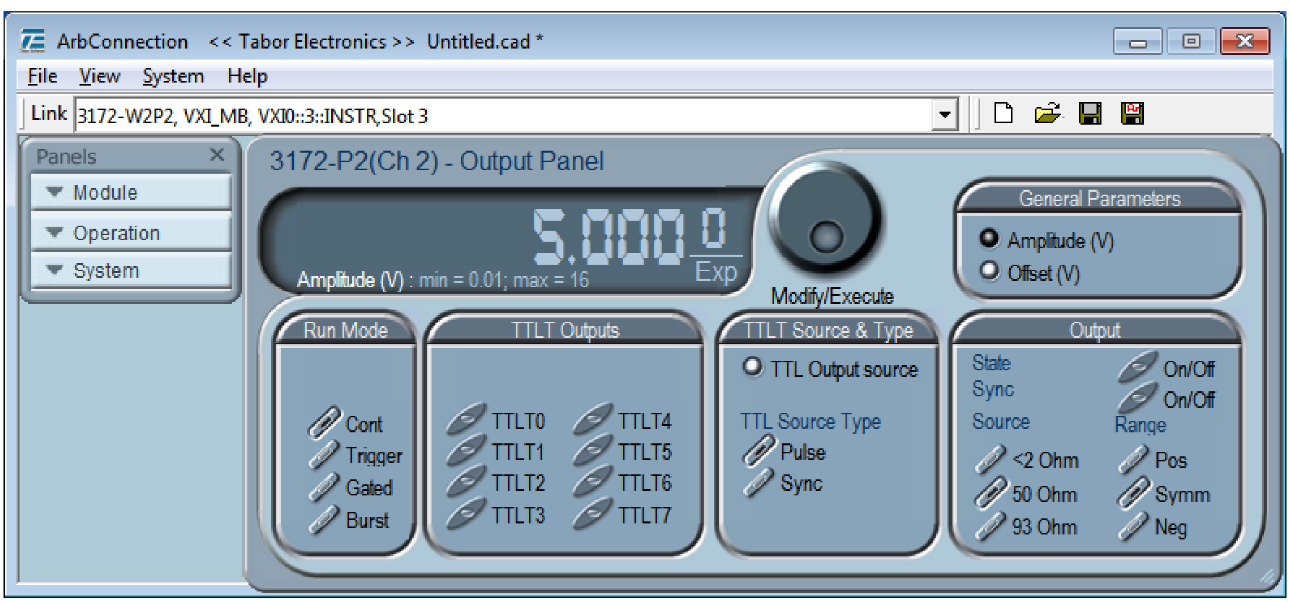

Figure 7-2, ArbConnection Main Window

5. If more than one VXI instrument is installed in the VXI chassis, select the one containing the module you wish to calibrate. Click the mouse where shown in Figure 7-3 to drop down a list of VXI instruments. Then select one by logical address. In Figure 7-3, only one VXI instrument is present.

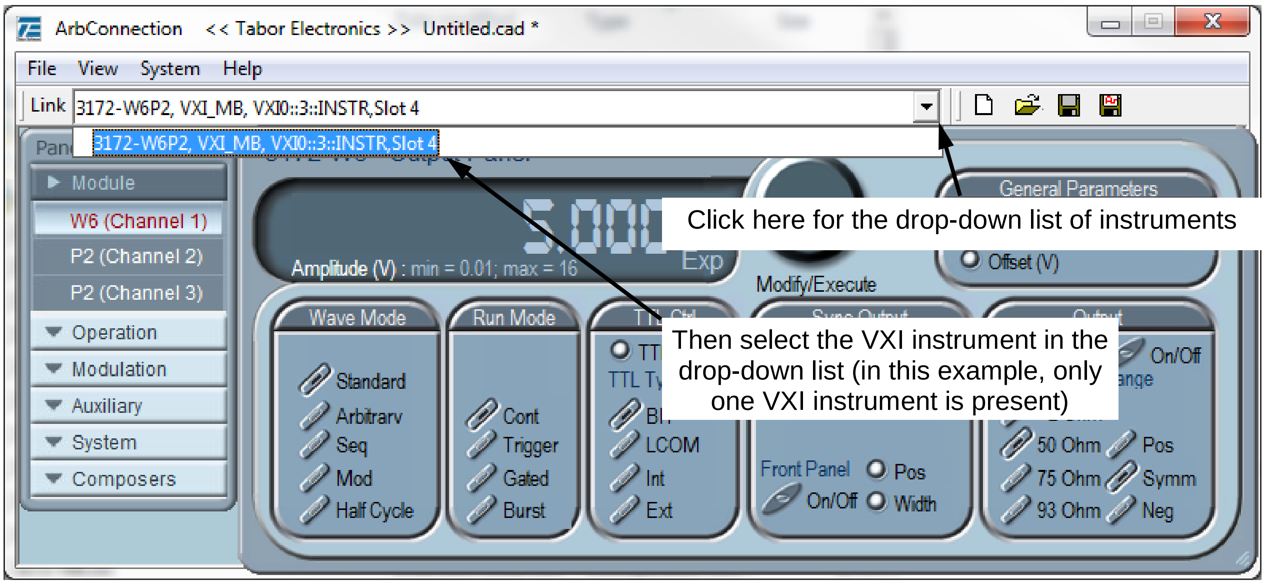

Figure 7-3, VXI Instrument Selection

Waveform Generator Calibration

After performing the setup and initialization in the above section, proceed with this section if you are calibrating a waveform generator module. If you are calibrating a pulse generator module, then go the Pulse Generator Calibration section below.

1. Using a coaxial cable (BNC male to Cinch or Cannon DM53740), connect the waveform generator front-panel output to the DMM input. See Figure 7-4 for front panel pin assignments.
2. In the ArbConnection window, click the Module button (Figure 7-5) to drop down the selection list. Then select the waveform generator channel to calibrate. Channels are numbered in ascending order, starting with the module installed in the upper position of the 3172 enclosure.

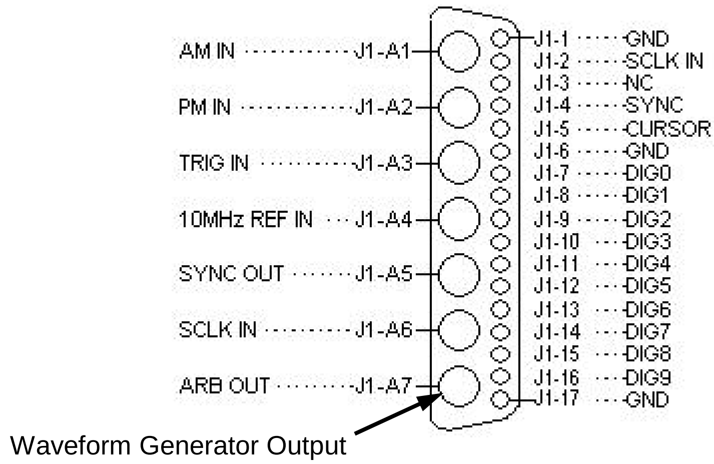

Figure 7-4, Front Panel Pin Assignments for Waveform Generator Module

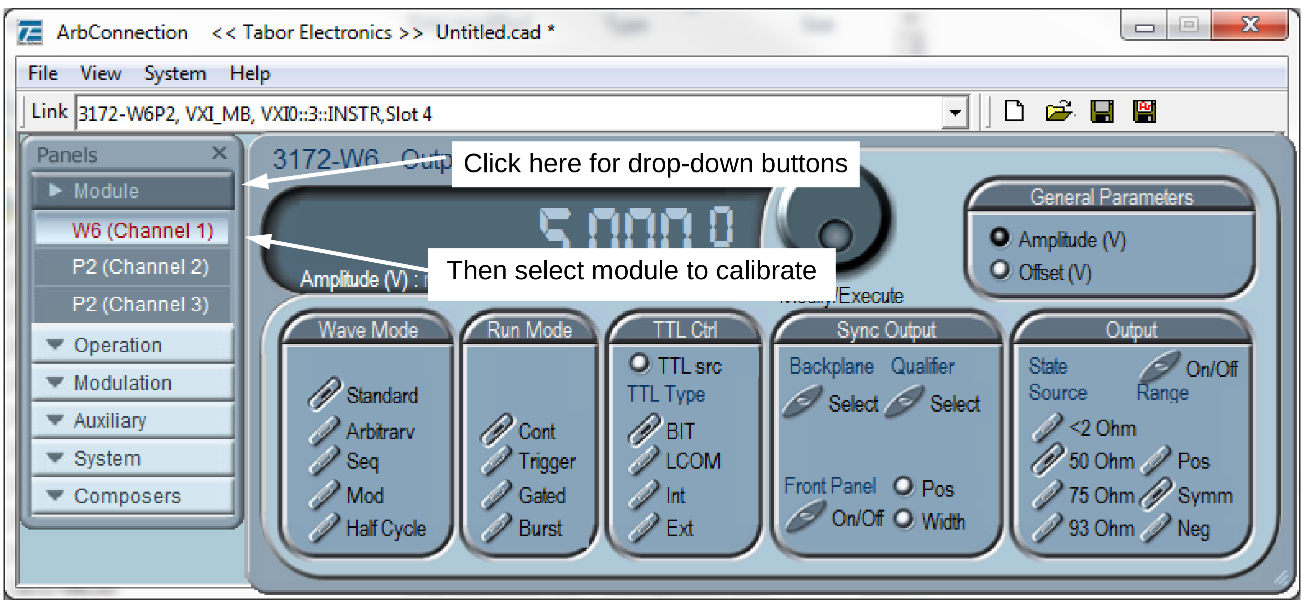

Figure 7-5, Selecting the Module to Calibrate

3. Click “System” (Figure 7-6). Then in the drop-down list, select “Calibration.” The Calibration Panel will then open (Figure 7-7).

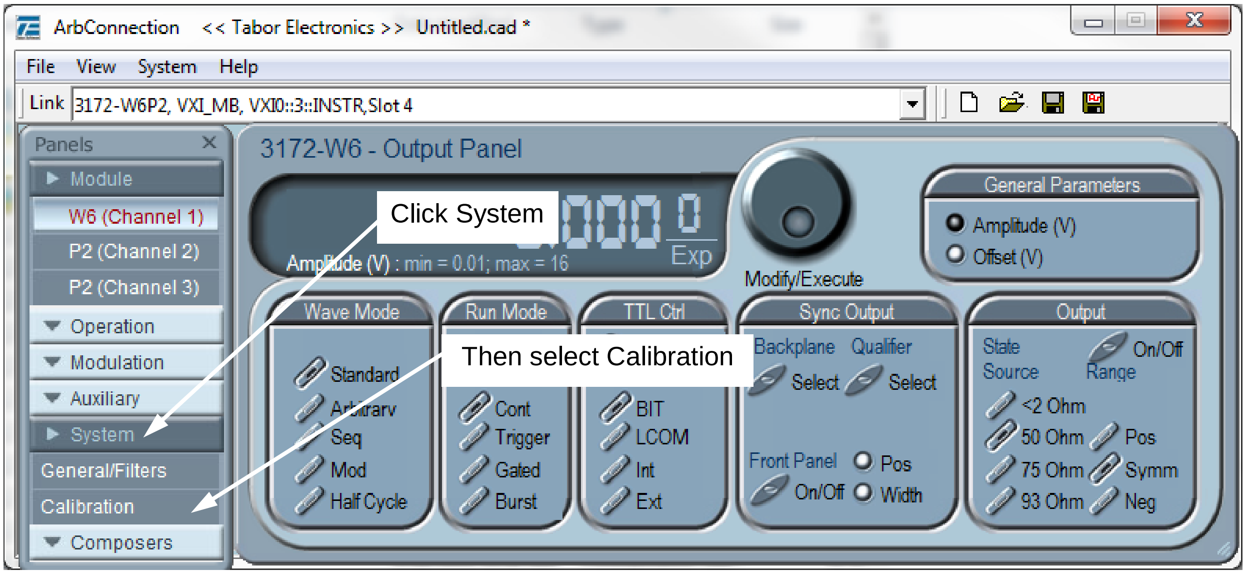

Figure 7-6, Selecting the Calibration Function

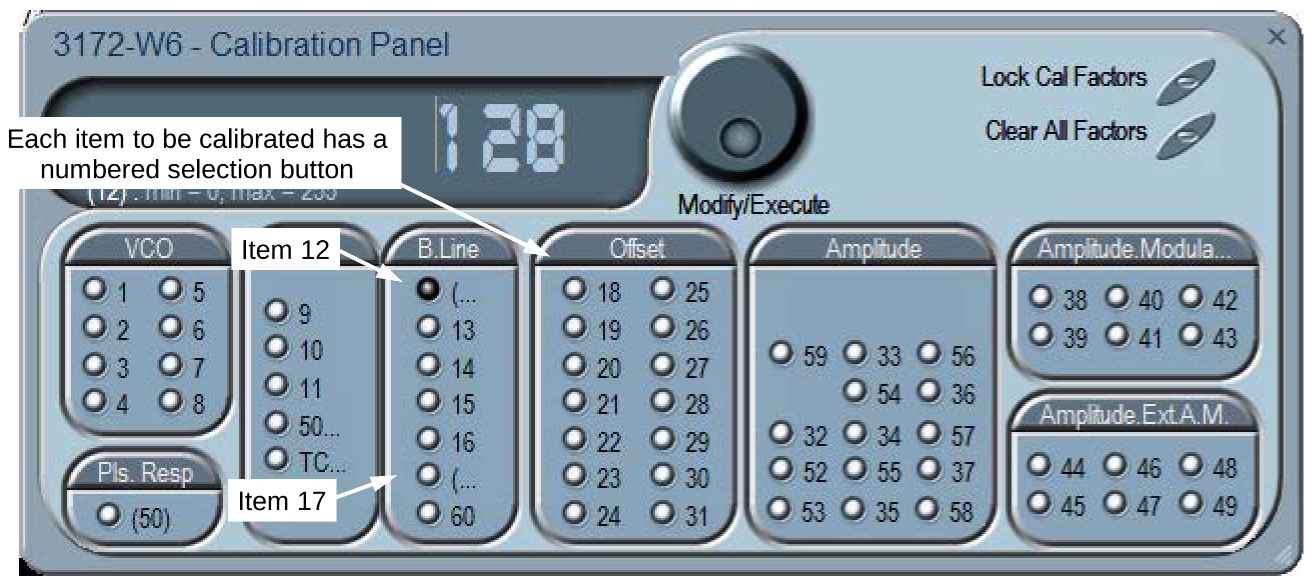

Figure 7-7, W6 Calibration Panel

4. In the Calibration Panel, note that each calibration item has a numbered selection button. For Periodic Calibration, only items 12 through 49 should be adjusted for W2 modules, and items 12 through 57 for W6 modules.

For each row in the appropriate calibration table below (Table 7-2 for W2 modules or Table 7-3 for W6 modules), do the following.

5. Verify that the DMM is set to the function (VDC or VAC) indicated in the calibration table.
6. In the Calibration Panel, select the calibration item (number)

indicated in the calibration table.

7. If a potentiometer is listed in the right-hand column of the calibration table, remove the right side panel and adjust the potentiometer to bring the DMM reading as close to the target value as feasible, and then skip to step 9. To keep the internal temperature stable, leave the side panel in place except when making potentiometer adjustments.

8. If no potentiometer is listed in the right column of the calibration table, then click the calibration adjustment number (Figure 7-8). Using the right arrow key on the computer keyboard, position the vertical cursor immediately to the left of the right-most digit as shown.

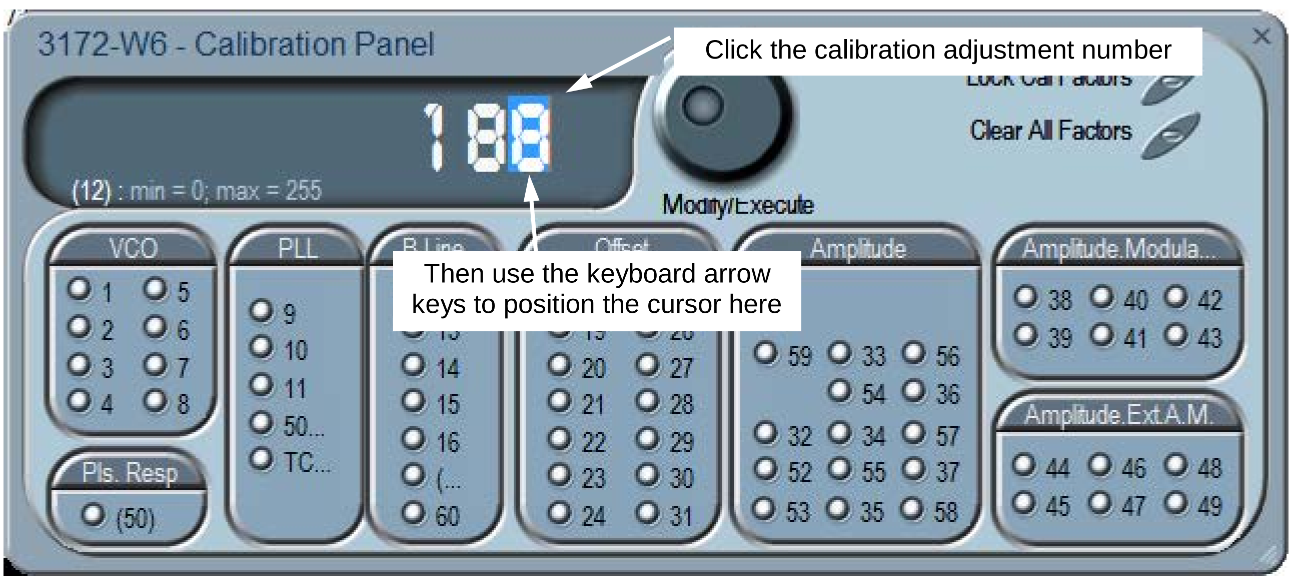

Figure 7-8, Making an Adjustment

9. Use the up-arrow and down-arrow keys on the computer keyboard to adjust the number up or down as required, while observing the reading on the DMM. Adjust the number until the DMM reading is as close as possible to the target value given in the calibration table.

10. When finished, verify that the DMM reading is within the low and high limits given in the table. If you are not able to calibrate within the specified range, there is either a problem with the way you are measuring the parameter or a problem with the 3172. In either case, do not leave any adjustment in its extreme setting but center the adjustment and contact your nearest service center for clarification and support.

11. Repeat steps 4 through 9 above for each row in the calibration table. Perform all steps in the order presented in the table.

12. Close the calibration panel by clicking the “x” in the upper-right corner.

Note
After making calibration adjustments, it is essential that you lock the calibration factors as described in the section titled “Locking the Calibration Factors” below. This will store the calibration factors in flash memory so that the instrument will retain its calibration when power is turned off.

Table 70, Calibration for W2 Module

Item in Cal Panel	DMM Function	Low Limit (Volts)	Target (Volts)	High Limit (Volts)	Adjust Pot
12	VDC	-0.020	0.00000	0.020	RV10
13	VDC	-0.020	0.00000	0.020	
14	VDC	-0.020	0.00000	0.020	
15	VDC	-0.020	0.00000	0.020	
16	VDC	-0.020	0.00000	0.020	
17	VDC	-0.050	0.00000	0.050	RV2
18	VDC	0.995	1	1.005	
19	VDC	2.490	2.5	2.510	
20	VDC	3.980	4	4.020	
21	VDC	5.475	5.5	5.525	
22	VDC	6.965	7	7.035	
23	VDC	8.460	8.5	8.540	
24	VDC	9.460	9.5	9.540	
25	VDC	-1.005	-1	-0.995	
26	VDC	-2.510	-2.5	-2.490	
27	VDC	-4.020	-4	-3.980	
28	VDC	-5.525	-5.5	-5.475	
29	VDC	-7.035	-7	-6.965	
30	VDC	-8.540	-8.5	-8.460	
31	VDC	-9.540	-9.5	-9.460	
Set DMM function to Volts AC					
32	VAC	1.7528	1.7678	1.7828	
33	VAC	0.52533	0.53033	0.53533	
34	VAC	0.17528	0.17678	0.17828	
35	VAC	0.052533	0.053033	0.053533	
36	VAC	0.017178	0.017678	0.018178	
37	VAC	0.0048033	0.0053033	0.0058033	
38	VAC	1.7528	1.7678	1.7828	
39	VAC	0.52533	0.53033	0.53533	
40	VAC	0.17528	0.17678	0.17828	
41	VAC	0.052533	0.053033	0.053533	
42	VAC	0.017178	0.017678	0.018178	
43	VAC	0.0048033	0.0053033	0.0058033	
44	VAC	1.7528	1.7678	1.7828	
45	VAC	0.52533	0.53033	0.53533	
46	VAC	0.17528	0.17678	0.17828	
47	VAC	0.052533	0.053033	0.053533	
48	VAC	0.017178	0.017678	0.018178	
49	VAC	0.0048033	0.0053033	0.0058033	
END					

Table 71, Calibration for W6 Module

Item in Cal Panel	DMM Function	Low Limit (Volts)	Target (Volts)	High Limit (Volts)	Adjust Pot
12	VDC	-0.020	0.00000	0.020	RV10
13	VDC	-0.020	0.00000	0.020	
14	VDC	-0.020	0.00000	0.020	
15	VDC	-0.020	0.00000	0.020	
16	VDC	-0.020	0.00000	0.020	
17	VDC	-0.050	0.00000	0.050	RV2
18	VDC	0.995	1	1.005	
19	VDC	2.490	2.5	2.510	
20	VDC	3.980	4	4.020	
21	VDC	5.475	5.5	5.525	
22	VDC	6.965	7	7.035	
23	VDC	8.460	8.5	8.540	
24	VDC	9.460	9.5	9.540	
25	VDC	-1.005	-1	-0.995	
26	VDC	-2.510	-2.5	-2.490	
27	VDC	-4.020	-4	-3.980	
28	VDC	-5.525	-5.5	-5.475	
29	VDC	-7.035	-7	-6.965	
30	VDC	-8.540	-8.5	-8.460	
31	VDC	-9.540	-9.5	-9.460	
Set DMM function to Volts AC					
59	VAC	3.8541	3.8891	3.9241	
32	VAC	3.8541	3.8891	3.9241	
52	VAC	2.4549	2.4749	2.4949	
53	VAC	1.3992	1.4142	1.4292	
33	VAC	1.1921	1.2021	1.2121	
54	VAC	0.41926	0.42426	0.42926	
34	VAC	0.34855	0.35355	0.35855	
55	VAC	0.13992	0.14142	0.14292	
35	VAC	0.10507	0.10607	0.10707	
56	VAC	0.041926	0.042426	0.042926	
36	VAC	0.034855	0.035355	0.035855	
57	VAC	0.013642	0.014142	0.014642	
37	VAC	0.010107	0.010607	0.011107	
58	VAC	0.001621	0.002121	0.002621	
38	VAC	1.7528	1.7678	1.7828	
39	VAC	0.52533	0.53033	0.53533	
40	VAC	0.17528	0.17678	0.17828	
41	VAC	0.052533	0.053033	0.053533	

Item in Cal Panel	DMM Function	Low Limit (Volts)	Target (Volts)	High Limit (Volts)	Adjust Pot
42	VAC	0.017178	0.017678	0.018178	
43	VAC	0.004803	0.005303	0.005803	
44	VAC	1.7528	1.7678	1.7828	
45	VAC	0.52533	0.53033	0.53533	
46	VAC	0.17528	0.17678	0.17828	
47	VAC	0.052533	0.053033	0.053533	
48	VAC	0.017178	0.017678	0.018178	
49	VAC	0.004803	0.005303	0.005803	
END					

Pulse Generator Calibration

This procedure calibrates the P2 pulse generator module. Each P2 module contains two channels. The procedure in this section must be performed separately for both channels.

1. Using a coaxial cable (BNC male to Cinch or Cannon DM53740), connect the pulse generator front-panel output to the DMM input. See Figure 7-9 for front panel pin assignments.
2. Click the Module button (Figure 7-10) to drop down the selection list. Then select the pulse generator channel to calibrate. Channels are numbered in ascending order, starting with the module installed in the upper position of the 3172 enclosure. The example shown in Figure 7-10 is for a 3172 having one W2 module (Channel 1) and one P2 module (PG1 is Channel 2, and PG2 is Channel 3).

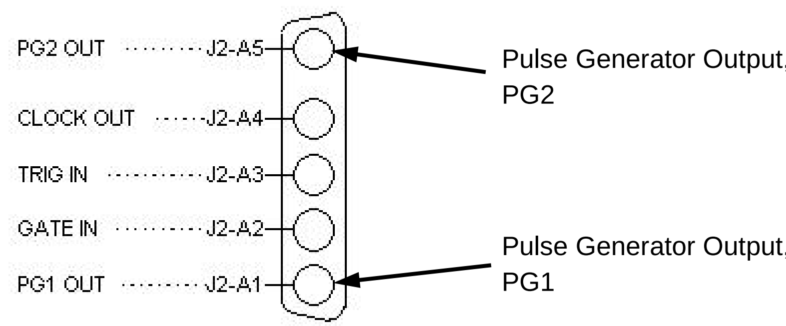

Figure 7-9, Front Panel Pin Assignments for Pulse Generator Module

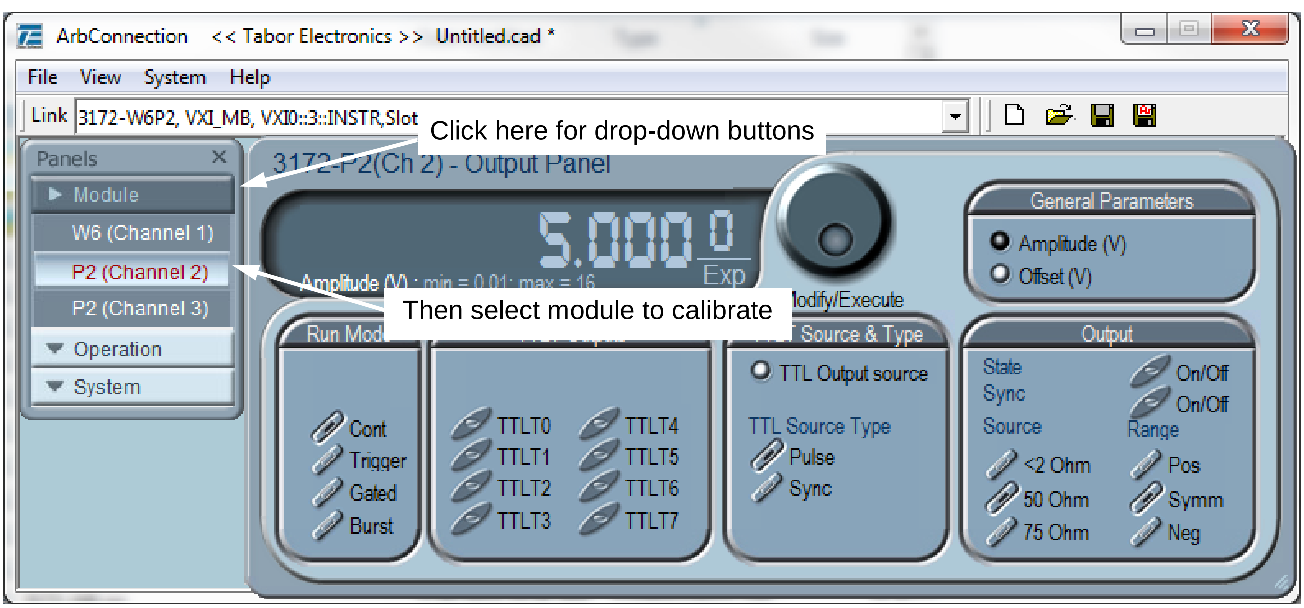

Figure 7-10, Selecting the Module to Calibrate

3. Click “System” (Figure 7-11). Then in the drop-down list, select “Calibration.” The Calibration Panel will then open (Figure 7-12).

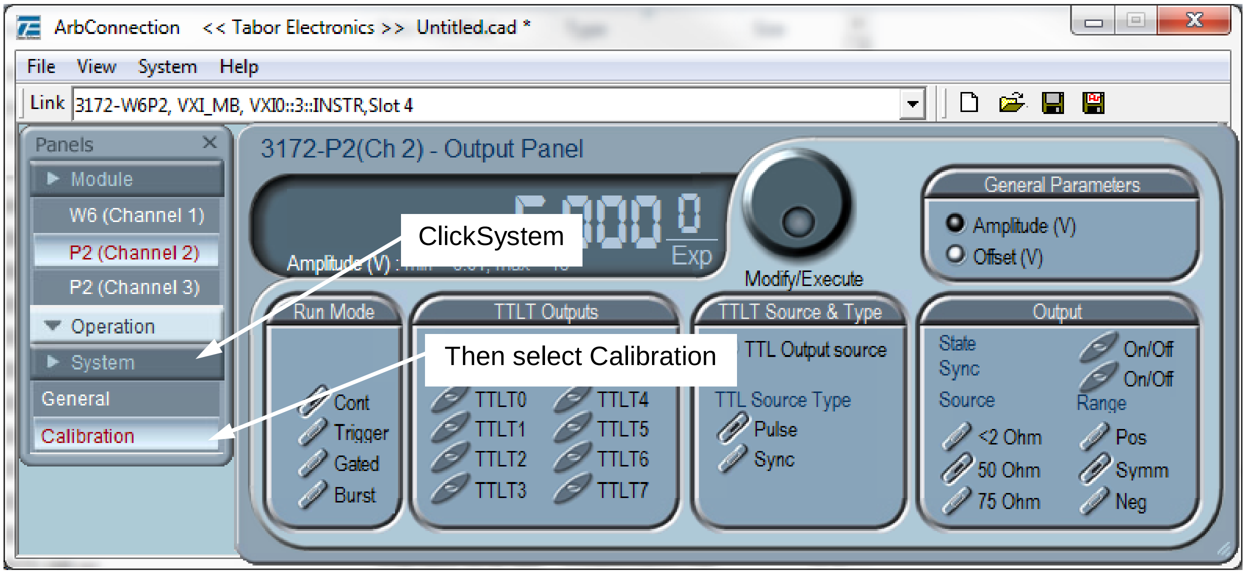

Figure 7-11, Selecting the Calibration Function

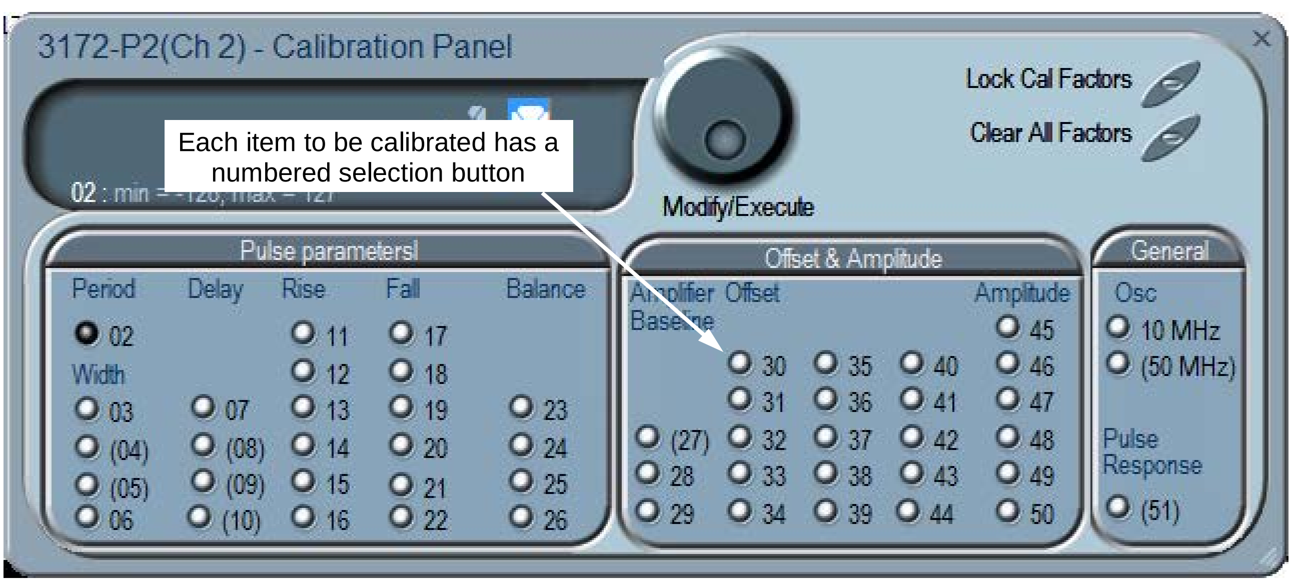

Figure 7-12, Calibration Panel

4. In the Calibration Panel, note that each calibration item has a numbered selection button. For Periodic Calibration, only items 27 through 50 should be adjusted.

For each row in Table 7-4 below, do the following.

5. Verify that the DMM is set to the function (VDC or VAC)

indicated in Table 7-4.

6. In the Calibration Panel, select the calibration item (number) indicated in Table 7-4.
7. If a potentiometer is listed in the right-hand column of Table 7-4, remove the right side panel and adjust the potentiometer to bring the DMM reading as close to the target value as feasible, and then skip to step 9. To keep the internal temperature stable, leave the side panel in place except when making potentiometer adjustments.
8. If no potentiometer is listed in the right column of the Table 7-4, then click the calibration adjustment number (Figure 7-13). Using the right arrow key on the computer keyboard, position the vertical cursor immediately to the left of the right-most digit as shown.

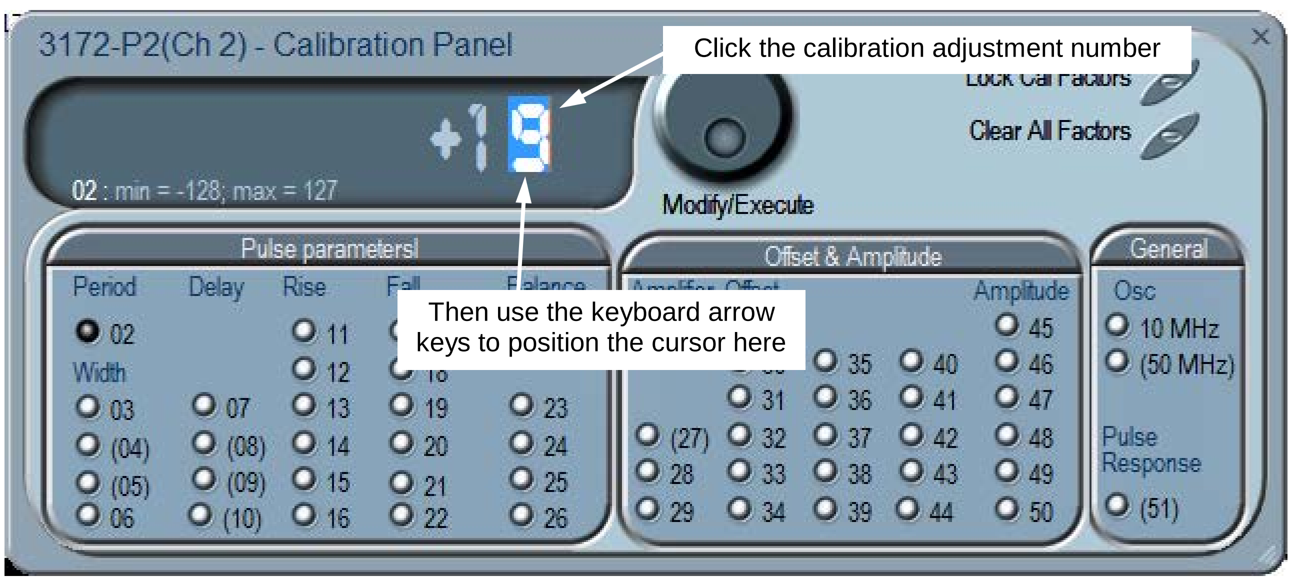

Figure 7-13, Making an Adjustment

9. Use the up-arrow and down-arrow keys on the computer keyboard to adjust the number up or down as required, while observing the reading on the DMM. Adjust the number until the DMM reading is as close as possible to the target value given in the calibration table.
10. When finished, verify that the DMM reading is within the low and high limits given in Table 7-4. If you are not able to calibrate within the specified range, there is either a problem with the way you are measuring the parameter or a problem with the 3172. In either case, do not leave any adjustment in its extreme setting but center the adjustment and contact your nearest service center for clarification and support.
11. Repeat steps 4 through 9 above for each row in Table 7-4.

Perform all steps in the order presented in the table.

12. Close the calibration panel by clicking the “x” in the upper-right corner.

Note
After making calibration adjustments, it is essential that you lock the calibration factors as described in the section titled “Locking the Calibration Factors” below. This will store the calibration factors in flash memory so that the instrument will retain its calibration when power is turned off.

Table 72, Calibration for P2 Module

Item in Cal Panel	DMM Function	Low Limit (Volts)	Target (Volts)	High Limit (Volts)	Adjust Pot
27	VDC	-0.020	0	0.020	Chan PG1: RV5* Chan PG2: RV2
28	VDC	-0.020	0	0.020	
29	VDC	-0.020	0	0.020	
Set DMM function to Volts AC					
30	VAC	-0.020	0	0.020	
Set DMM function to Volts DC					
31	VDC	0.995	1	1.005	
32	VDC	2.490	2.5	2.510	
33	VDC	3.980	4	4.020	
34	VDC	5.475	5.5	5.525	
35	VDC	6.965	7	7.035	
36	VDC	8.460	8.5	8.540	
37	VDC	9.460	9.5	9.540	
38	VDC	-1.005	-1	-0.995	
39	VDC	-2.510	-2.5	-2.490	
40	VDC	-4.020	-4	-3.980	
41	VDC	-5.525	-5.5	-5.475	
42	VDC	-7.035	-7	-6.965	
43	VDC	-8.540	-8.5	-8.460	
44	VDC	-9.540	-9.5	-9.460	
Set DMM function to Volts AC					
45	VAC	2.485	2.5	2.515	
46	VAC	0.745	0.75	0.755	
47	VAC	0.2485	0.250	0.2515	
48	VAC	0.0745	0.075	0.0755	
49	VAC	0.0245	0.025	0.0255	
50	VAC	0.0070	0.0075	0.0080	
END					

* Each P2 module has two output channels: PG1 and PG2. When calibrating PG1, adjust potentiometer RV5. When calibrating PG2, adjust potentiometer RV2.

Locking the Calibration Factors

After all items have been calibrated, lock the calibration factors as follows.

1. Click the mouse on the "Lock Cal Factors" button in the Calibration Panel (Figure 7-14).

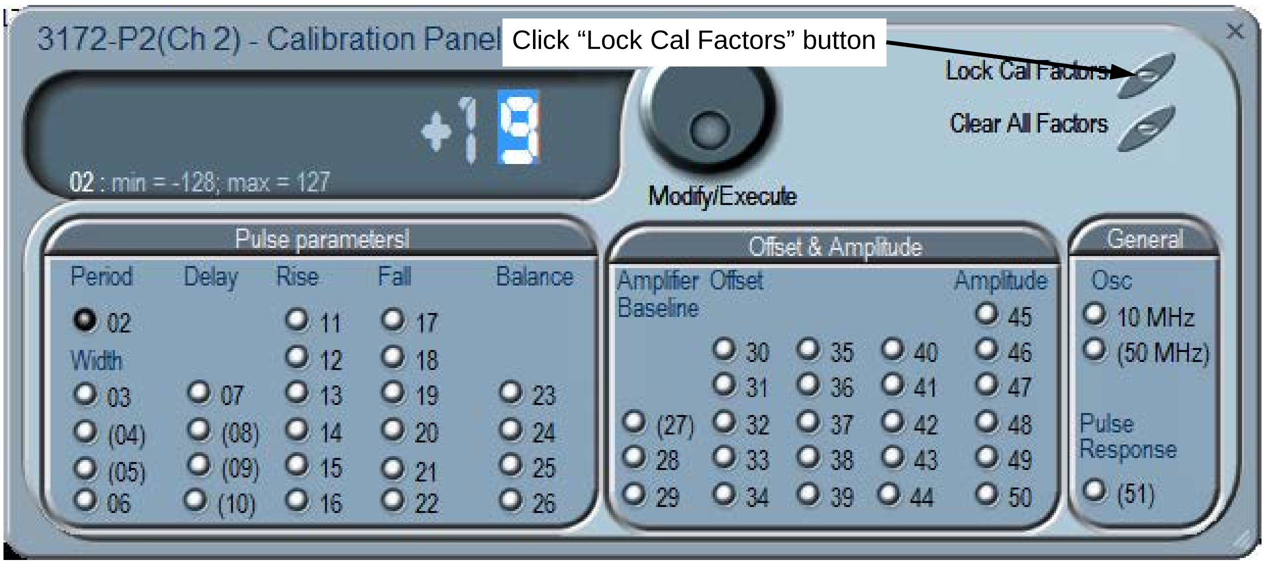

Figure 7-14, Locking the Calibration Factors

2. A pop-up window will appear (Figure 7-15).

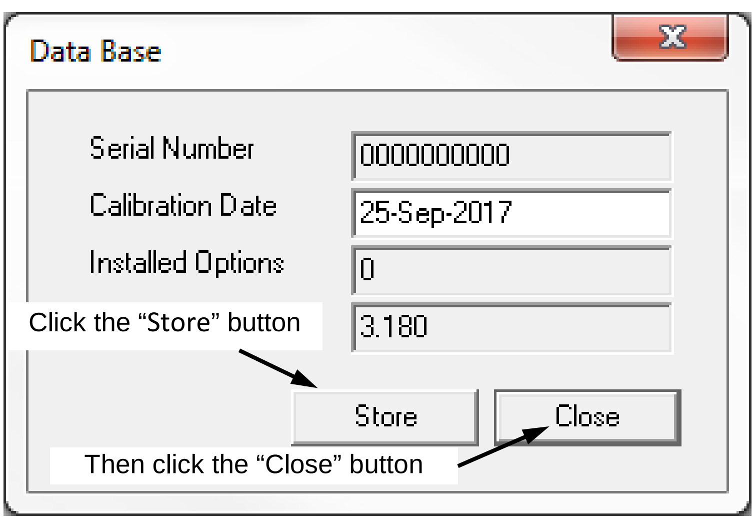

Figure 7-15, Confirmation for Locking Calibration Factors

3. Click the “Store” button. Since the computer screen gives no visual response to the store operation, you may wish to get a visual indication by observing a brief flash of the front-panel “Link” indicator as you click the “Store” button.
4. Click the “Close” button.
5. Exit the ArbConnection software by clicking the “X” in the upper right-hand corner of the main window. This completes the calibration procedure.

Factory Calibration

Factory calibration is performed with the 3172 installed in a VXIbus chassis. Either a register-based or message-based controller may be used, and the procedure is the same for both options. The ArbConnection software application must be installed and interfaced to the instrument.

Some calibration steps require removal of the right side panel (cover). To keep the internal temperature stable, leave the cover on except while making adjustments that require its removal. To facilitate removal while the unit is in the VXI chassis, remove all except two screws before installing the unit. Replace all screws after calibration is complete.

Warm-up

Some calibration steps require removal of the right side panel (cover). To keep the internal temperature stable, leave the cover on during warm-up and calibration, except while making adjustments that require its removal. To facilitate removal while the unit is in the VXI chassis, remove all except two screws before installing the unit. Replace all screws after calibration is complete.

Install the 3172 in a VXIbus chassis. Turn on chassis power and allow the 3172 to warm up and stabilize for at least 30 minutes before calibrating. If the instrument has been subjected to temperatures outside the range of 25°C ± 5°C, allow at least one additional hour for the instrument to stabilize before calibrating.

Procedure, W6 Module

1. Invoke ArbConnection.
2. Click the Module button in the Panels bar. Select the channel to calibrate.
3. Click the System button in the Panels bar. Then click Calibration. The Calibration Panel (Figure 7-16) will appear.

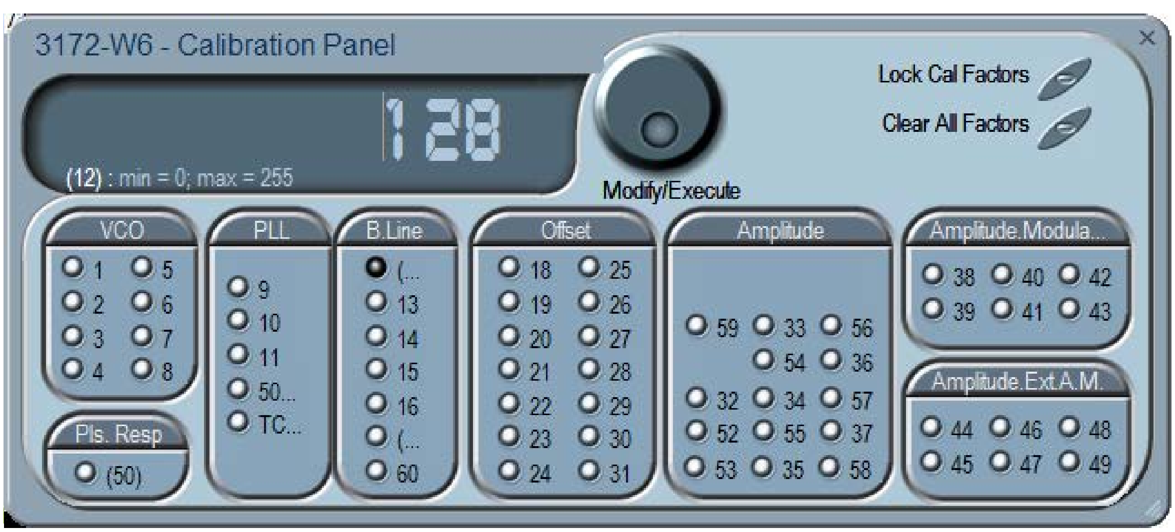

Figure 7-16, Calibration Panel

Note
Parameters that are adjusted only during Factory Calibration are enclosed in parentheses. All other parameters are calibrated during Factory Calibration or Periodic Calibration.

In the panel, each parameter to be calibrated is labeled with a number from 1 to 58 except the 50MHz and TCXO adjustments in the PLL group. Adjustments should be carried out exactly in sequential order. The numbers that are associated with each adjustment are identified by a setup number in the title of each of the adjustments in the following procedure.

Adjustment values have a range of 0 through 255, with a center setting of 128. If you are not able to calibrate within the specified range, there is either a problem with the way you are measuring the parameter or a problem with the instrument. In either case, do not leave any adjustment in its extreme setting but center the adjustment and resolve the problem before proceeding with the calibration.

Note that in the following procedures, although configuration of the 3172 is done automatically, configuration details are shown for reference. There is no requirement to change the 3172 configuration during the adjustment procedure except where specifically noted.

Note
After making calibration adjustments, it is essential that you lock the calibration factors as described in the section “Locking the Calibration Factors” at the end of this procedure. This stores the calibration factors in flash memory so that the instrument will retain its calibration after power is turned off.

VCO Adjustments

The VCO controls the accuracy and linearity of the sample clock generator. Correct operation of the VCO circuit assures the accuracy of the frequency path. Use this procedure if you suspect that there is a frequency linearity and/or accuracy issue.

Setup 1

160 MHz SCLK

Equipment: DMM

Preparation:

1. Configure the DMM as follows:

Function: DCV
Range: 1 V

2. Configure the 3172 as follows:
   SCLK: 160 MHz
3. Connect the DMM probes between R107 and ground

Adjustment:

1. Adjust CAL:SETup 2 for a DMM reading of 0 V, ±100 mV.

Setup 2

180 MHz SCLK

Equipment: DMM

Preparation:

1. Configure the DMM as follows:
   Function: DCV
   Range: 1 V
2. Configure the 3172 as follows:
   SCLK: 180 MHz
3. Connect the DMM probes between R107 and ground

Adjustment:

1. Adjust CAL:SETup 3 for a DMM reading of 0 V, ±100 mV.

Setup 3

200 MHz SCLK

Equipment: DMM

Preparation:

1. Configure the DMM as follows:
   Function: DCV
   Range: 1 V
2. Configure the 3172 as follows:
   SCLK: 200 MHz
3. Connect the DMM probes between R107 and ground

Adjustment:

1. Adjust CAL:SETup 4 for a DMM reading of 0 V, ±100 mV.

Setup 4

110 MHz SCLK

Equipment: DMM

Preparation:

1. Configure the DMM as follows:
   Function: DCV
   Range: 1 V
2. Configure the 3172 as follows:
   SCLK: 110 MHz
3. Connect the DMM probes between R107 and ground

Adjustment:

1. Adjust CAL:SETup 5 for a DMM reading of 0 V, ±100 mV.

Setup 5

120 MHz SCLK

Equipment: DMM

Preparation:

1. Configure the DMM as follows:
   Function: DCV
   Range: 1 V
2. Configure the 3172 as follows:
   SCLK: 120 MHz
3. Connect the DMM probes between R107 and ground

Adjustment:

1. Adjust CAL:SETup 6 for a DMM reading of 0 V, ±100 mV.

Setup 6

130 MHz SCLK

Equipment: DMM

Preparation:

1. Configure the DMM as follows:
   Function: DCV
   Range: 1 V
2. Configure the 3172 as follows:
   SCLK: 130 MHz
3. Connect the DMM probes between R107 and ground

Adjustment:

1. Adjust CAL:SETup 7 for a DMM reading of 0 V, ±100 mV.

Setup 7

139 MHz SCLK

Equipment: DMM

Preparation:

1. Configure the DMM as follows:
   Function: DCV
   Range: 1 V
2. Configure the 3172 as follows:
   SCLK: 139MHz
3. Connect the DMM probes between R107 and ground

Adjustment:

1. Adjust CAL:SETup 8 for a DMM reading of 0 V, ±100 mV.

Setup 8

145 MHz SCLK

Equipment: DMM

Preparation:

1. Configure the DMM as follows:

Function: DCV
Range: 1 V

2. Configure the 3172 as follows:
   SCLK: 145 MHz
3. Connect the DMM probes between R107 and ground

Adjustment:

1. Adjust CAL:SETup 9 for a DMM reading of 0 V, ±100 mV

PLL Adjustments

The PLL operation is basic to the sample clock generator. Correct operation of the PLL circuit assures the accuracy of the frequency path as well as the accuracy of the PLL phase. Use this procedure if you suspect that there is a frequency and/or PLL accuracy issue.

Setup 9

PLL Preparation

Equipment: No equipment required for this step.

Preparation:

1. Configure the 3172 as follows:
   Function Mode: Arbitrary
   Wavelength: 1000 points
   Waveform: Square
   Run Mode: PLL
2. Set CAL:SETUP 61
3. Set CAL:VAL 120
4. Set CAL:SETUP 63

Adjustment: None required. This step is part of the preparation for Setup 10 below.

Setup 10

Phase “0”

Equipment: Digital Oscilloscope, Function Generator, BNC "T" Connector (one male to two females), BNC to BNC coaxial cable

Preparation:

1. Prepare the external function generator waveform as follows:
   Amplitude: 5 Vp-p
   Waveform: Square
   Frequency: 10 kHz
2. Set the oscilloscope channel impedance as follows:
   Channel 1: 1 MΩ
   Channel 2: 50 Ω
3. Connect a BNC “T” connector to oscilloscope channel 1.
4. Connect one end of the “T” connector to the function generator output.
5. Connect the other end of the “T” connector to the TRG/PLL input of the 3172.
6. Connect the 3172 output to oscilloscope channel 2.
7. Configure the 3172 as follows:
   Function Mode: Arbitrary

Wavelength: 1000 points
Waveform: Square
Run Mode: PLL

Adjustment:

8. Adjust CAL:SETUP 62 for counter reading of -250 ns to 250 ns.

Setup 11

Phase Fine “0”

Equipment: Counter, Function Generator, "T" Connector, BNC to BNC cables

Preparation:

1. Prepare the external function generator waveform as follows:
   Amplitude: 5 Vp-p
   Waveform: Square
   Frequency: 10 kHz
2. Configure the counter as follows:
   Channel 1 input impedance: High
   Channel 2 input impedance: 50 Ω
   Function: TI A -> B
3. Connect a BNC “T” connector to counter channel A
4. Connect one end of the “T” connector to the function generator output.
5. Connect the other end of the “T” connector to the TRG/PLL input of the 3172, using a 50 Ω terminator at the 3172 connector.
6. Connect the 3172 output to counter input B.
7. Configure the 3172 as follows:
   Function Mode: Arbitrary
   Wavelength: 1000 points
   Waveform: Square
   Run Mode: PLL
   Fine Phase Offset: +30°

Adjustment:

1. Adjust CAL:SETUP 61 for a counter reading of 8.28 μs to 8.38 μs.
2. Repeat Setup 10 and Setup 11 until the errors have been minimized.

Reference Oscillators Adjustments

Use this procedure to adjust the reference oscillators. The reference oscillators determine the accuracy of the output frequency so if you suspect that there is an accuracy issue, proceed with the calibration of the reference oscillators. Note that the 50MHz is marked as a factory adjustment and therefore, it is not normally required to be performed during normal calibration cycles except if the gated oscillator accuracy does not meet the published specification limits, or after a repair has been executed on this same circuit.

(Setup 50MHz)

50 MHz Gated Oscillator Adjustment

Equipment: Counter, Function Generator, BNC to BNC cables
Preparation:

1. Configure the counter as follows:
   Termination: 50 Ω DC
   Function: TI A -> B
   Slope B: Negative
2. Connect the 3172 output to the oscilloscope input
3. Connect an external function generator to the front panel TRG/PLL connector
4. Using ArbConnection prepare and download the following waveform:
   Wavelength: 100 points
   Waveform: Pulse: Delay = 0.01 %,
   Rise/Fall Time = 0 %, High Time = 99.99 %
5. Configure the 3172 as follows:
   Function: Arbitrary
   Run Mode: Triggered
   Retrigger Mode: On
   Retrigger Delay: 20 μs
   Output: On
6. Using an external function generator, manually trigger the 3172

Adjustment:

1. Set CAL:SET 0
2. Adjust C10 for a period of 20 μs, ±5 %

Setup TCXO

10 MHz TCXO Frequency

Equipment: Counter, BNC to BNC cables
Preparation:

1. Configure the counter as follows:
   Function: Freq A
   Termination: 50 Ω
2. Connect the 3172 output to the counter input.
3. Configure the 3172 as follows:
   Frequency: 10 MHz
   Ref Source: Internal
   Output: On
   Amplitude 2 V
   Waveform: Square

Adjustment:

1. Adjust CAL:SETup1 for counter reading of 10 MHz, ±2 Hz.

Base Line Offset Adjustments

The base line offset adjustments assure that the AC signal is symmetrical around the 0V line. Use this procedure if you suspect that there is a base line accuracy issue.

Setup 12

Null Offset Symmetrical voltage range

Equipment: DMM, BNC to BNC cable, 50 Ω feed through termination, Dual banana to BNC adapter

Preparation:

1. Configure the DMM as follows:
   Function: DCV
   Range: 100 mV
2. Connect the 3172 output to the DMM input.
   Terminate the 3172 output at the DMM input with the, 50 Ω feed through termination
3. Configure the 3172 as follows:
   Function Standard
   Waveform Sine
   Voltage Range Symmetrical
   Amplitude 10 mV
   Output: On

Adjustment:

1. Set CAL:SETup 13
2. Adjust RV10 for DMM reading of 0 V, ±20 mV.

Setup 13

Amplifier Offset Positive voltage range

Equipment: DMM, BNC to BNC cable, 50 Ω feed through termination, Dual banana to BNC adapter

Preparation:

1. Configure the DMM as follows:
   Function: DCV
   Range: 100 mV
2. Connect the 3172 output to the DMM input. Terminate the 3172 output at the DMM input with the, 50 Ω feedthrough termination
3. Configure the 3172 as follows:
   Voltage Range Positive
   Amplitude 10 mV
   Output: On

Adjustment:

1. Adjust CAL:SETUP 14 for DMM reading of 0 V, ±20 mV.

Setup 14

Amplifier Offset Negative voltage range

Equipment: DMM, BNC to BNC cable, 50 Ω feed through termination, Dual banana to BNC adapter

Preparation:

1. Configure the DMM as follows:
   Function: DCV
   Range: 100 mV
2. Connect the 3172 output to the DMM input. Terminate the 3172 output at the DMM input with the, 50 Ω feed through termination
3. Configure the 3172 as follows:
   Voltage Range Negative
   Amplitude -10 mV
   Output: On

Adjustment:

1. Adjust CAL:SETup 15 for DMM reading of 0 V, ±20 mV.

Setup 15

Base Line Offset – Modulation

Equipment: DMM, BNC to BNC cable, 50 Ω feed through termination, Dual banana to BNC adapter

Preparation:

1. Configure the DMM as follows:
   Function: DCV
   Range: 100 mV
2. Connect the 3172 output to the DMM input. Terminate the 3172 output at the DMM input with the, 50 Ω feed through termination
3. Configure the 3172 as follows:
   Function: Modulated
   Output: On

Adjustment:

1. Adjust CAL:SETup 16 for DMM reading of 0 V, ±20 mV.

Setup 16

Base Line Offset – Arbitrary

Equipment: DMM, BNC to BNC cable, 50 Ω feedthrough termination, Dual banana to BNC adapter

Preparation:

1. Configure the DMM as follows:
   Function: DCV
   Range: 100 mV
2. Connect the 3172 output to the DMM input. Terminate the 3172 output at the DMM input with the, 50 Ω feedthrough termination
3. Configure the 3172 as follows:
   Output: On

Adjustment:

1. Adjust CAL:SETup 17 for DMM reading of 0 V, ±20 mV

Setup 17

Base Line Offset – External AM

Equipment: DMM, BNC to BNC cable, 50 Ω feed through termination, Dual banana to BNC adapter

Preparation:

1. Configure the DMM as follows:
   Function: DCV
   Range: 100 mV
2. Connect the 3172 Ext. AM input to -10 VDC Voltage Source
3. Connect the 3172 output to the DMM input. Terminate the 3172 output at the DMM input with the, 50 Ω feed through termination
4. Configure the 3172 as follows:
   Function: External AM
   Output: On

Adjustment:

1. Set CAL:SETup 51
2. Adjust RV2 for DMM reading of 0 V, ±50 mV

Offset Adjustments

The offset adjustments assure that the DC offsets are within the specified range. Use this procedure if you suspect that the offset accuracy is an issue.

Setup 18

+ 1 V Offset Output

Equipment: DMM, BNC to BNC cable, 50 Ω feed through termination, Dual banana to BNC adapter

Preparation:

1. Configure the DMM as follows:
   Function: DCV
   Range: 1 V
2. Connect the 3172 output to the DMM input. Terminate the 3172 output at the DMM input with the 50 Ω feed through termination
3. Configure the 3172 as follows:
   Voltage Range: Positive
   Amplitude: 10 mV
   Offset +0.995 V
   Output: On

Adjustment:

1. Adjust CAL: SETup 18 for DMM reading of +1 V, ± 5 mV

Setup 19

+2.5 V Offset Output

Equipment: DMM, BNC to BNC cable, 50 Ω feed through termination, Dual banana to BNC adapter

Preparation:

1. Configure the DMM as follows:
   Function: DCV

Range: 10 V

2. Connect the 3172 output to the DMM input. Terminate the 3172 output at the DMM input with the 50 Ω feed through termination
3. Configure the 3172 as follows:

   Voltage Range: Positive
   Amplitude: 10 mV
   Offset +2.495 V
   Output: On

Adjustment:

1. Adjust CAL: SETup 19 for DMM reading of +2.5 V, ± 10 mV

Setup 20

+4 V Offset Output

Equipment: DMM, BNC to BNC cable, 50 Ω feed through termination, Dual banana to BNC adapter

Preparation:

1. Configure the DMM as follows:

   Function: DCV
   Range: 10 V
2. Connect the 3172 output to the DMM input. Terminate the 3172 output at the DMM input with the 50 Ω feed through termination
3. Configure the 3172 as follows:

   Voltage Range: Positive
   Amplitude: 10 mV
   Offset +3.995 V
   Output: On

Adjustment:

1. Adjust CAL: SETup 20 for DMM reading of +4 V, ± 20 mV

Setup 21

+5.5 V Offset Output

Equipment: DMM, BNC to BNC cable, 50 Ω feed through termination, Dual banana to BNC adapter

Preparation:

1. Configure the DMM as follows:

   Function: DCV
   Range: 20V
2. Connect the 3172 output to the DMM input. Terminate the 3172 output at the DMM input with the 50 Ω feed through termination
3. Configure the 3172 as follows:

   Voltage Range: Positive
   Amplitude: 10 mV
   Offset +5.495 V
   Output: On

Adjustment:

1. Adjust CAL: SETup 21 for DMM reading of +5.5 V, ± 25 mV

Setup 22

+7V Offset Output

Equipment: DMM, BNC to BNC cable, 50 Ω feed through termination, Dual

banana to BNC adapter

Preparation:

1. Configure the DMM as follows:
   Function: DCV
   Range: 20 V
2. Connect the 3172 output to the DMM input. Terminate the 3172 output at the DMM input with the 50 Ω feed through termination
3. Configure the 3172 as follows:
   Voltage Range: Positive
   Amplitude: 10 mV
   Offset +6.995 V
   Output: On

Adjustment:

1. Adjust CAL: SETup 22 for DMM reading of +7 V, ± 35 mV

Setup 23

+8.5 V Offset Output

Equipment: DMM, BNC to BNC cable, 50 Ω feed through termination, Dual banana to BNC adapter

Preparation:

1. Configure the DMM as follows:
   Function: DCV
   Range: 20V
2. Connect the 3172 output to the DMM input. Terminate the 3172 output at the DMM input with the 50 Ω feed through termination
3. Configure the 3172 as follows:
   Voltage Range: Positive
   Amplitude: 10 mV
   Offset +8.495 V
   Output: On

Adjustment:

1. Adjust CAL: SETup 23 for DMM reading of +8.5 V, ± 40 mV

Setup 24

+9.5 V Offset Output

Equipment: DMM, BNC to BNC cable, 50 Ω feed through termination, Dual banana to BNC adapter

Preparation:

1. Configure the DMM as follows:
   Function: DCV
   Range: 10 V
2. Connect the 3172 output to the DMM input. Terminate the 3172 output at the DMM input with the 50 Ω feed through termination
3. Configure the 3172 as follows:
   Voltage Range: Positive
   Amplitude: 10 mV
   Offset +9.495 V
   Output: On

Adjustment:

1. Adjust CAL: SETup 24 for DMM reading of +9.5 V, ± 40 mV

Setup 25

-1 V Offset Output

Equipment: DMM, BNC to BNC cable, 50 Ω feed through termination, Dual banana to BNC adapter

Preparation:

1. Configure the DMM as follows:
   Function: DCV
   Range: 1 V
2. Connect the 3172 output to the DMM input. Terminate the 3172 output at the DMM input with the 50 Ω feed through termination
3. Configure the 3172 as follows:
   Voltage Range: Negative
   Amplitude: -10 mV
   Offset -0.995 V
   Output: On

Adjustment:

1. Adjust CAL: SETup 25 for DMM reading of -1 V, ± 5 mV

Setup 26

-5 V Offset Output

Equipment: DMM, BNC to BNC cable, 50 Ω feed through termination, Dual banana to BNC adapter

Preparation:

1. Configure the DMM as follows:
   Function: DCV
   Range: 10 V
2. Connect the 3172 output to the DMM input. Terminate the 3172 output at the DMM input with the 50 Ω feed through termination
3. Configure the 3172 as follows:
   Voltage Range: Negative
   Amplitude: -10 mV
   Offset -2.495 V
   Output: On

Adjustment:

1. Adjust CAL: SETup 26 for DMM reading of -2.55 V, ± 10 mV

Setup 27

-4 V Offset Output

Equipment: DMM, BNC to BNC cable, 50 Ω feed through termination, Dual banana to BNC adapter

Preparation:

1. Configure the DMM as follows:
   Function: DCV
   Range: 10 V
2. Connect the 3172 output to the DMM input. Terminate the 3172 output at the DMM input with the 50 Ω feed through termination
3. Configure the 3172 as follows:
   Voltage Range: Negative
   Amplitude: -10 mV
   Offset -3.995V

Output: On

Adjustment:

1. Adjust CAL: SETup 27 for DMM reading of -4 V, ± 20 mV.

Setup 28

-5.5 V Offset Output

Equipment: DMM, BNC to BNC cable, 50 Ω feed through termination, Dual banana to BNC adapter

Preparation:

1. Configure the DMM as follows:
   Function: DCV
   Range: 10 V
2. Connect the 3172 output to the DMM input. Terminate the 3172 output at the DMM input with the 50 Ω feed through termination
3. Configure the 3172 as follows:
   Voltage Range: Negative
   Amplitude: -10 mV
   Offset -5.495 V
   Output: On

Adjustment:

1. Adjust CAL: SETup 28 for DMM reading of -5.5 V, ± 25 mV.

Setup 29

-7 V Offset Output

Equipment: DMM, BNC to BNC cable, 50 Ω feed through termination, Dual banana to BNC adapter

Preparation:

1. Configure the DMM as follows:
   Function: DCV
   Range: 10 V
2. Connect the 3172 output to the DMM input. Terminate the 3172 output at the DMM input with the 50 Ω feed through termination
3. Configure the 3172 as follows:
   Voltage Range: Negative
   Amplitude: -10 mV
   Offset -6.995 V
   Impedance 50 ohms
   Output: On

Adjustment:

1. Adjust CAL: SETup 29 for DMM reading of -7 V, ± 35 mV

Setup 30

-8.5 V Offset Output

Equipment: DMM, BNC to BNC cable, 50 Ω feed through termination, Dual banana to BNC adapter

Preparation:

1. Configure the DMM as follows:
   Function: DCV
   Range: 10 V

2. Connect the 3172 output to the DMM input. Terminate the 3172 output at the DMM input with the 50 Ω feed through termination
3. Configure the 3172 as follows:
   Voltage Range: Negative
   Amplitude: -10 mV
   Offset -8.495 V
   Output: On

Adjustment:
1. Adjust CAL: SETup 30 for DMM reading of -8.5 V, ± 40 mV

Setup 31

-9.5 V Offset Output

Equipment: DMM, BNC to BNC cable, 50 Ω feed through termination, Dual banana to BNC adapter

Preparation:
1. Configure the DMM as follows:
   Function: DCV
   Range: 10 V
2. Connect the 3172 output to the DMM input. Terminate the 3172 output at the DMM input with the 50 Ω feed through termination
3. Configure the 3172 as follows:
   Voltage Range: Negative
   Amplitude: -10 mV
   Offset -9.495 V
   Output: On

Adjustment:
1. Adjust CAL: SETup 31 for DMM reading of -9.5 V, ± 40 mV**.**

Amplitude Adjustments

The amplitude adjustments assure that the AC levels are within the specified range. Use this procedure if you suspect that the amplitude accuracy is an issue.

Setup 32

5 V Amplitude - Arbitrary

Equipment: DMM, BNC to BNC cable, 50 Ω feed through termination, Dual banana to BNC adapter

Preparation:
1. Configure the DMM as follows:
   Function: ACV
   Range: 10 V
2. Connect the 3172 output to the DMM input. Terminate the 3172 output at the DMM input with the, 50 Ω feed through termination
3. Configure the 3172 as follows:
   Frequency: 1 kHz
   Output: On

Adjustment:
1. Adjust CAL:SETup 32 for DMM reading of 1.768 V ±15 mV

Setup 33

1.5 V Amplitude - Arbitrary

Equipment: DMM, BNC to BNC cable, 50 Ω feed through termination, Dual banana to BNC adapter

Preparation:

1. Configure the DMM as follows:
   Function: ACV
   Range: 10 V
2. Connect the 3172 output to the DMM input. Terminate the 3172 output at the DMM input with the, 50 Ω feed through termination.
3. Configure the 3172 as follows:
   Frequency: 1 kHz
   Output: On
   Amplitude: 1.5 V

Adjustment:

1. Adjust CAL:SETup 33 for DMM reading of 530mV ± 5 mV

Setup 34

1 V Amplitude - Arbitrary

Equipment: DMM, BNC to BNC cable, 50 Ω feed through termination, Dual banana to BNC adapter

Preparation:

1. Configure the DMM as follows:
   Function: ACV
   Range: 1 V
2. Connect the 3172 output to the DMM input. Terminate the 3172 output at the DMM input with the, 50 Ω feed through termination.
3. Configure the 3172 as follows:
   Frequency: 1 kHz
   Output: On
   Amplitude: 0.5 V

Adjustment:

1. Adjust CAL:SETup 34 for DMM reading of 176.8 mV ± 1.5mV

Setup 35

150 mV Amplitude – Arbitrary

Equipment: DMM, BNC to BNC cable, 50 Ω feed through termination, Dual banana to BNC adapter

Preparation:

1. Configure the DMM as follows:
   Function: ACV
   Range: 1 V
2. Connect the 3172 output to the DMM input. Terminate the 3172 output at the DMM input with the, 50 Ω feed through termination.
3. Configure the 3172 as follows:
   Frequency: 1 kHz
   Output: On
   Amplitude: 150 mV

Adjustment:

1. Adjust CAL:SETup 35 for DMM reading of 53.1mV ± 0.5mV

Setup 36

50 mV Amplitude - Arbitrary

Equipment: DMM, BNC to BNC cable, 50 Ω feed through termination, Dual banana to BNC adapter

Preparation:

1. Configure the DMM as follows:
   Function: ACV
   Range: 100 mV
2. Connect the 3172 output to the DMM input. Terminate the 3172 output at the DMM input with the, 50 Ω feed through termination
3. Configure the 3172 as follows:
   Frequency: 1 kHz
   Output: On
   Amplitude: 50 mV

Adjustment:

1. Adjust CAL:SETup 36 for DMM reading of 17.68 mV ±0.5 mV

Setup 37

15 mV Amplitude – Arbitrary

Equipment: DMM, BNC to BNC cable, 50 Ω feed through termination, Dual banana to BNC adapter

Preparation:

1. Configure the DMM as follows:
   Function: ACV
   Range: 100 mV
2. Connect the 3172 output to the DMM input. Terminate the 3172 output at the DMM input with the, 50 Ω feed through termination.
3. Configure the 3172 as follows:
   Frequency: 1 kHz
   Output: On
   Amplitude: 15 mV

Adjustment:

1. Adjust CAL:SETup 37 for DMM reading of 5.31mV ± 0.5mV

Amplitude Adjustments-Modulation

The amplitude adjustments assure that the AC levels are within the specified range. Use this procedure if you suspect that the amplitude accuracy is an issue.

Setup 38

5 V Amplitude - Modulation

Equipment: DMM, BNC to BNC cable, 50 Ω feed through termination, Dual banana to BNC adapter

Preparation:

1. Configure the DMM as follows:
   Function: ACV
   Range: 10 V

2. Connect the 3172 output to the DMM input. Terminate the 3172 output at the DMM input with the, 50 Ω feed through termination.
3. Configure the 3172 as follows:
   Function: Modulated
   Frequency: 1 kHz
   Output: On
   Amplitude: 5 V

Adjustment:
1. Adjust CAL:SETup 38 for DMM reading of 1.768 V ± 15 mV

Setup 39

1.5 V Amplitude - Modulation

Equipment: DMM, BNC to BNC cable, 50 Ω feed through termination, Dual banana to BNC adapter

Preparation:
1. Configure the DMM as follows:
   Function: ACV
   Range: 10 V
2. Connect the 3172 output to the DMM input. Terminate the 3172 output at the DMM input with the, 50 Ω feed through termination.
3. Configure the 3172 as follows:
   Function: Modulated
   Frequency: 1 kHz
   Output: On
   Amplitude: 1.5 V

Adjustment:
1. Adjust CAL:SETup 39 for DMM reading of 530mV ± 5 mV

Setup 40

0.5 V Amplitude – Modulation

Equipment: DMM, BNC to BNC cable, 50 Ω feed through termination, Dual banana to BNC adapter

Preparation:
1. Configure the DMM as follows:
   Function: ACV
   Range: 1 V
2. Connect the 3172 output to the DMM input. Terminate the 3172 output at the DMM input with the, 50 Ω feed through termination.
3. Configure the 3172 as follows:
   Function: Modulated
   Frequency: 1 kHz
   Output: On
   Amplitude: 0.5 V

Adjustment:
1. Adjust CAL:SETup 40 for DMM reading of 176.5 mV ± 1.5mV

Setup 41

150 mV Amplitude - Modulation

Equipment: DMM, BNC to BNC cable, 50 Ω feed through termination, Dual

banana to BNC adapter

Preparation:

1. Configure the DMM as follows:
   - Function: ACV
   - Range: 1 V
2. Connect the 3172 output to the DMM input. Terminate the 3172 output at the DMM input with the, 50 Ω feed through termination.
3. Configure the 3172 as follows:
   - Function: Modulated
   - Frequency: 1 kHz
   - Output: On
   - Amplitude: 150 mV

Adjustment:

1. Adjust CAL:SETup 41for DMM reading of 53.1 mV ±0.5 mV

Setup 42

50 mV Amplitude - Modulation

Equipment: DMM, BNC to BNC cable, 50 Ω feed through termination, Dual banana to BNC adapter

Preparation:

1. Configure the DMM as follows:
   - Function: ACV
   - Range: 100 mV
2. Connect the 3172 output to the DMM input. Terminate the 3172 output at the DMM input with the, 50 Ω feed through termination.
3. Configure the 3172 as follows:
   - Function: Modulated
   - Frequency: 1 kHz
   - Output: On
   - Impedance 50 Ohms
   - Amplitude: 100 mV

Adjustment:

1. Adjust CAL:SETup 42 for DMM reading of 17.68 mV ±0.5 mV

Setup 43

15 mV Amplitude - Modulation

Equipment: DMM, BNC to BNC cable, 50 Ω feed through termination, Dual banana to BNC adapter

Preparation:

1. Configure the DMM as follows:
   - Function: ACV
   - Range: 100 mV
2. Connect the 3172 output to the DMM input. Terminate the 3172 output at the DMM input with the, 50 Ω feed through termination.
3. Configure the 3172 as follows:
   - Function: Modulated
   - Frequency: 1 kHz
   - Output: On
   - Amplitude: 15 mV

Adjustment:

1. Adjust CAL:SETup 43 for DMM reading of 5.31mV ±0.5 mV

Amplitude Adjustments - External AM

The adjustments assure that the amplitude levels are within the specified range. Use this procedure if you suspect that the amplitude modulation accuracy is an issue.

Setup 44

5 V Amplitude - External AM

Equipment: DMM, BNC to BNC cable, 50 Ω feed through termination, Dual banana to BNC adapter

Preparation:

1. Configure the DMM as follows:
   Function: ACV
   Range: 10 V
2. Connect the 3172 output to the DMM input. Terminate the 3172 output at the DMM input with the, 50 Ω feed through termination.
3. Configure the 3172 as follows:
   Frequency: 1 kHz
   Output: On
   Amplitude: 5V
   Function: Arbitrary
   External AM ON

Adjustment:

1. Adjust CAL:SETup 44 for DMM reading of 1.768 V ± 15 mV

Setup 45

1.5 V Amplitude - External AM

Equipment: DMM, BNC to BNC cable, 50 Ω feed through termination, Dual banana to BNC adapter

Preparation:

1. Configure the DMM as follows:
   Function: ACV
   Range: 10 V
2. Connect the 3172 output to the DMM input. Terminate the 3172 output at the DMM input with the, 50 Ω feed through termination.
3. Configure the 3172 as follows:
   Frequency: 1 kHz
   Output: On
   Amplitude: 1.5 V
   Function: Arbitrary
   External AM ON

Adjustment:

1. Adjust CAL:SETup 45 for DMM reading of 530 mV ±5 mV

Setup 46

0.5 V Amplitude - External AM

Equipment: DMM, BNC to BNC cable, 50 Ω feed through termination, Dual

banana to BNC adapter

Preparation:

1. Configure the DMM as follows:
   Function: ACV
   Range: 1 V

2. Connect the 3172 output to the DMM input. Terminate the 3172 output at the DMM input with the, 50 Ω feed through termination.

3. Configure the 3172 as follows:
   Frequency: 1 kHz
   Output: On
   Function: Arbitrary
   External AM ON
   Amplitude: 0.5 V
   Offset: 0 V

Adjustment:

1. Adjust CAL:SETup 46 for DMM reading of 176.8mV ±1.5 mV

Setup 47 150 mV Amplitude - External AM

Equipment: DMM, BNC to BNC cable, 50 Ω feed through termination, Dual banana to BNC adapter

Preparation:

1. Configure the DMM as follows:
   Function: ACV
   Range: 1 V
2. Connect the 3172 output to the DMM input. Terminate the 3172 output at the DMM input with the, 50 Ω feed through termination.
3. Configure the 3172 as follows:
   Frequency: 1 kHz
   Output: On
   Function: Arbitrary
   External AM ON
   Amplitude: 150 mV

Adjustment:

1. Adjust CAL:SETup 47 for DMM reading of 53.1 mV ±0.5 mV

Setup 48 50 mV Amplitude - External AM

Equipment: DMM, BNC to BNC cable, 50 Ω feed through termination, Dual banana to BNC adapter

Preparation:

1. Configure the DMM as follows:
   Function: ACV
   Range: 100 mV
2. Connect the 3172 output to the DMM input. Terminate the 3172 output at the DMM input with the, 50 Ω feed through termination.
3. Configure the 3172 as follows:
   Frequency: 1 kHz
   Output: On
   Function: Arbitrary

External AM ON
Amplitude: 50 mV

Adjustment:

1. Adjust CAL:SETup 48 for DMM reading of 17.68 mV ±0.5 mV

Setup 49

15 mV Amplitude - External AM

Equipment: DMM, BNC to BNC cable, 50 Ω feed through termination, Dual banana to BNC adapter

Preparation:

1. Configure the DMM as follows:
   Function: ACV
   Range: 100 mV
2. Connect the 3172 output to the DMM input. Terminate the 3172 output at the DMM input with the, 50 Ω feed through termination.
3. Configure the 3172 as follows:
   Frequency: 1 kHz
   Output: On
   Function: Arbitrary
   External AM ON
   Amplitude: 15 mV

Adjustment:

1. Adjust CAL:SETup 49 for DMM reading of 5.31mV ±0.5 mV

Pulse Response Adjustments

The pulse response adjustments assure that the rise and fall times, as well as, the aberrations are within the specified range. Use this procedure if you suspect that the pulse response is an issue. Note that setups 48 and 49 are marked as a factory adjustment and therefore, it is not normally required to be performed during normal calibration cycles except if the pulse response of the output stage has been degraded and does not meet the published specification limits, or after a repair has been executed on these very circuit.

(Setup 50)

Pulse Response

Mechanical calibration

Equipment: Oscilloscope, BNC to BNC cable, 20 dB feed through attenuator

Preparation:

1. Configure the 3172 as follows:
   Function: Square
   Amplitude: 6V
2. Connect the 3172 output to the oscilloscope input. Set oscilloscope input impedance to 50 Ω.
3. Set oscilloscope vertical sensitivity to 20 mV

Adjustment:

1. Set CAL:SETup 50
2. Adjust vertical trace to 6 divisions

3. Adjust RV1 and RV3 for best pulse response (8 ns type, 5% aberrations)

Procedure, P2 Module

1. Invoke ArbConnection.
2. Click the Module button in the Panels bar. Select the channel to calibrate.
3. Click the System button in the Panels bar. Then click Calibration. The Calibration Panel (Figure 7-17) will appear.

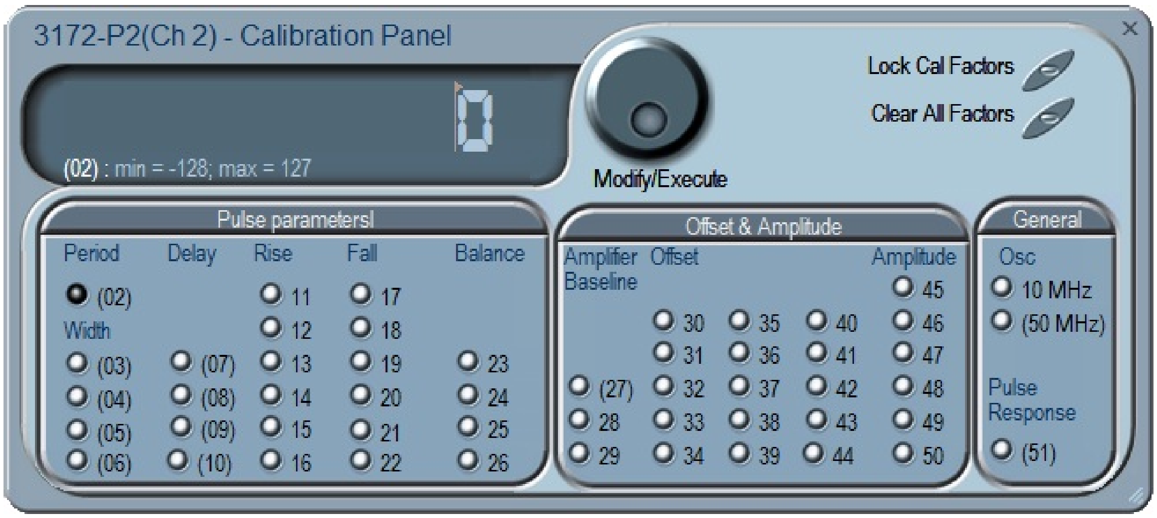

Figure 7-17, P2, Pulse Generator Channel 1 Calibration Panel

Note
Parameters that are adjusted only during Factory Calibration are enclosed in parentheses. All other parameters are calibrated during Factory Calibration or Periodic Calibration.

Calibrations are marked with numbers from 1 to 51 and should be carried out exactly in the order as numbered on the panel. The numbers that are associated with each adjustment are identified as Setup Number at the title of each of the adjustments in the following procedure.

Remote adjustments have the range of 1 through 256 with the center alignment set to 128. Therefore, if you are not sure of the direction, set the adjustment to 128 and add or subtract from this value. If you have reached 1 or 256 and were not able to calibrate the range, there is either a problem with the way you measure the parameter or possibly there is a problem with the instrument. In either case, do not leave any adjustment in its extreme setting but center the adjustment and contact your nearest service center for clarifications and support.

Note in the following procedures that although configuration of the P2 is done automatically, some of the configurations are shown for reference only. There is no requirement to change configuration of the P2 during the remote adjustment procedure except in places where specifically noted.

Reference Oscillator Adjustments

Use this procedure to adjust the reference oscillators. The reference oscillators determine the accuracy of the output period so if you suspect that there is an accuracy issue; proceed with the calibration of the reference oscillators. Note that the 50MHz is marked as a factory adjustment and therefore, it is not normally required to be performed during normal calibration cycles except if the gated oscillator accuracy does not meet the published specification limits, or after a repair has been executed on this same circuit.

Setup 10 MHz

10 MHz TCXO Frequency

Equipment: Counter
Preparation:

1. Configure the counter as follows:
   Function: Freq A
   Termination: 50 Ω
2. Connect the P2 output to the counter input.
3. Configure the P2 as follows:
   Frequency: 10 MHz
   Output: On
   Amplitude 2 V
   Waveform: Square

Adjustment:

1. Adjust CAL:SETup 1 for counter reading of 10 MHz, ±2 Hz.

(Setup 50MHz)

50 MHz Gated Oscillator Adjustment

Equipment: Counter, Function Generator, BNC to BNC cables
Preparation:

1. Configure the counter as follows:
   Termination: 50 Ω DC
   Function: TI A -> B
2. Configure the function generator as follows:
   Waveform: Square wave
   Frequency: 1 kHz
   Amplitude: 1 V
   Offset: 1 V
3. Place a “T” connector on the function generator output and connect one end to the P2 TRIG input and the other end to the A channel on the counter.
4. Connect the P2 output to the B channel of the counter.

5. Configure the P2 as follows:

Period:	1 µs
Run Mode:	Triggered
Trigger Delay:	On
Delay:	10 µs
Output:	On

Adjustment:
1. Program CAL:SET 0
2. Adjust C351 for a period of 10 µs, ±5 %.

Pulse Period Adjustments

The pulse period adjustments assure that the period of the pulse generator is within the specified limits. Note that clock generator is separate for each channel and therefore, after you do the adjustments on channel 1 continue with the same adjustments on channel 2. Use this procedure if you suspect that the pulse period is not within range.

Setup 02

Pulse Period

Equipment: Counter
Preparation:
1. Configure the counter as follows:

Function:	Period A
Termination:	50 Ω

2. Connect the P2 output to the counter input.
3. Configure the P2 as follows:

Output:	On
Waveform:	Square

Adjustment:
1. Adjust CAL:SETup 2 for counter reading of 1 ms, ±1 µs.

Pulse Width Adjustments

The pulse width adjustments assure that the widths of the generated pulses are within the specified limits. Note that each channel has its own width adjustments and therefore, after you do the adjustments on channel 1 continue with the same adjustments on channel 2. Use this procedure if you suspect that the pulse width is not within range.

(Setup 03)

100 µs Pulse Width

Equipment: Counter, Oscilloscope

Preparation:
1. Configure the counter as follows:

Function:	Pulse A
Termination:	50 Ω

2. Connect the P2 output to the counter input.
3. Configure the P2 as follows:

Output: On

Adjustment:

1. Adjust CAL:SETup 3 for counter reading of 100 µs, ±0.1 µs.
2. Note and record the value of setup 3 and calculate K0 using the following equation. K0 will be required the pulse width balance adjustments in Setups 04, 05 and 06.

   K0 = 1/[1+(setup 3 reading + 128)/1000]

   Calculation example:
   For a setup 3 reading = -15
   K0 = 1/[1+(-15+128)/1000]=0.89847

3. Connect an oscilloscope probe on U473, pin 3 (U415, pin 3 in channel 2).
4. Adjust RV6 (RV3 in channel 2) for vertical symmetry about the 0 V line.

(Setups 04 & 05)

Pulse Width Balance

Equipment: Oscilloscope

Preparation:

1. Configure the oscilloscope as follows:
   Vertical: 1 V/div
   Horizontal: 5 ns/div
   Termination: 50 Ω
   Measure: Pulse width
2. Connect the P2 output to the oscilloscope input.
3. Configure the P2 as follows:
   Output: On

Adjustment:

1. Program CAL:SETup 4 and note and record the reading in units of ns. Note as PWA.
2. Program CAL:SETup 5 and note and record the reading in units of ns. Note as PWB.
3. Select the best value for R116 (R47 in channel 2) for the following relationship:

   PWB = (PWA-10xK0), ±0.4 ns

   Note: use K0 from Setup 03

(Setup 06)

Low Range Pulse Width Balance

Equipment: Oscilloscope

Preparation:

1. Configure the oscilloscope as follows:
   Vertical: 1 V/div
   Horizontal: 5 ns/div
   Termination: 50 Ω
   Measure: Pulse width
2. Connect the P2 output to the oscilloscope input.

3. Configure the P2 as follows:
   Output: On

Adjustment:

1. Adjust CAL:SETup 6 for pulse width reading of 10xK0, ±0.4 ns
   Note: use K0 from Setup 03

Pulse Delay Adjustments

The pulse delay adjustments assure that the delays of the generated pulses are within the specified limits. Note that each channel has its own delay adjustments and therefore, after you do the adjustments on channel 1 continue with the same adjustments on channel 2. Use this procedure if you suspect that the pulse delay is not within range.

(Setup 07)

100 µs Delay

Equipment: Counter

Preparation:

1. Configure the counter as follows:
   Function: period A
   Termination: 50 Ω
2. Connect the P2 output to the counter input.
3. Configure the P2 as follows:
   Output: On

Adjustment:

1. Adjust CAL:SETup 7 for counter reading of 100 µs, ±0.1 µs.
2. Note and record the value of setup 7 and calculate K1 using the following equation. K0 will be required the delay balance adjustments in Setups 08, 09 and 10.

K1 = 1/[1+(setup 7 reading + 128)/1000]

Calculation example:
For a setup 7 reading = 64
K1 = 1/[1+(64+128)/1000]=0.8389

(Setups 08)

Delay Balance

Equipment: Oscilloscope

Preparation:

1. Configure the oscilloscope as follows:
   Vertical: 1 V/div
   Horizontal: 5 ns/div
   Termination: 50 Ω
   Measure: Time interval (sync to output)
2. Connect the P2 output to the oscilloscope input.
3. Connect the P2 sync output to the oscilloscope. Synchronize the oscilloscope on the sync output.
4. Configure the P2 as follows:

Output: On

Adjustment:

1. Program CAL:SETup 8 and note and record the reading in units of ns. Note as DLYA.

(Setup 10)

Zero Delay Compensation

Equipment: Oscilloscope

Preparation:

1. Configure the oscilloscope as follows:
   - Vertical: 1 V/div
   - Horizontal: 5 ns/div
   - Termination: 50 Ω
   - Measure: Pulse width
2. Connect the P2 output to the oscilloscope input.
3. Connect the P2 sync output to the oscilloscope. Synchronize the oscilloscope on the sync output.
4. Configure the P2 as follows:
   - Output: On

Adjustment:

1. Program CAL:SETup 10
2. Adjust C37 (C15 in channel 2) for the same DLYB reading as in step 08, ±0.4 ns

Where DLYB = (DLYA-20xK1), ±0.2 ns

Note: use K1 from Setup 07

Pulse Leading Edge Adjustments

The pulse leading edge adjustments assure that the leading edge transition times of the generated pulses are within the specified limits. Note that each channel has its own leading edge adjustments and therefore, after you do the adjustments on channel 1 continue with the same adjustments on channel 2. Use this procedure if you suspect that the leading edge timing is not within range.

Setup 11

50 ns Leading Edge

Equipment: oscilloscope

Preparation:

1. Configure the oscilloscope as follows:
   - Vertical: 1 V/div
   - Horizontal: 10 ns/div
   - Termination: 50 Ω
   - Measure: Rise time
2. Connect the P2 output to the oscilloscope input.

3. Connect the P2 sync output to the oscilloscope. Synchronize the oscilloscope on the sync output.
4. Configure the P2 as follows:
   Output: On

Adjustment:
1. Adjust CAL:SETup 11 for a rise time reading of 50 ns, ±2 ns.

Setup 12

200 ns Leading Edge

Equipment: oscilloscope

Preparation:
1. Configure the oscilloscope as follows:
   Vertical: 1 V/div
   Horizontal: 50 ns/div
   Termination: 50 Ω
   Measure: Rise time
2. Connect the P2 output to the oscilloscope input.
3. Connect the P2 sync output to the oscilloscope. Synchronize the oscilloscope on the sync output.
4. Configure the P2 as follows:
   Output: On

Adjustment:
1. Adjust CAL:SETup 12 for a rise time reading of 200 ns, ±10 ns.

Setup 13

2 μs Leading Edge

Equipment: oscilloscope

Preparation:
1. Configure the oscilloscope as follows:
   Vertical: 1 V/div
   Horizontal: 500 ns/div
   Termination: 50 Ω
   Measure: Rise time
2. Connect the P2 output to the oscilloscope input.
3. Connect the P2 sync output to the oscilloscope. Synchronize the oscilloscope on the sync output.
4. Configure the P2 as follows:
   Output: On

Adjustment:
1. Adjust CAL:SETup 13 for a rise time reading of 2 μs, ±50 ns.

Setup 14

20 μs Leading Edge

Equipment: oscilloscope

Preparation:

1. Configure the oscilloscope as follows:
   Vertical: 1 V/div
   Horizontal: 5 µs/div
   Termination: 50 Ω
   Measure: Rise time
2. Connect the P2 output to the oscilloscope input.
3. Connect the P2 sync output to the oscilloscope. Synchronize the oscilloscope on the sync output.
4. Configure the P2 as follows:
   Output: On

Adjustment:
1. Adjust CAL:SETup 14 for a rise time reading of 20 µs, ±0.5 µs.

Setup 15

200 µs Leading Edge

Equipment: oscilloscope

Preparation:
1. Configure the oscilloscope as follows:
   Vertical: 1 V/div
   Horizontal: 50 µs /div
   Termination: 50 Ω
   Measure: Rise time
2. Connect the P2 output to the oscilloscope input.
3. Connect the P2 sync output to the oscilloscope. Synchronize the oscilloscope on the sync output.
4. Configure the P2 as follows:
   Output: On

Adjustment:
1. Adjust CAL:SETup 15 for a rise time reading of 200 µs, ±5 µs.

Setup 15

2 ms Leading Edge

Equipment: oscilloscope

Preparation:
1. Configure the oscilloscope as follows:
   Vertical: 1 V/div
   Horizontal: 0.5 ms/div
   Termination: 50 Ω
   Measure: Rise time
2. Connect the P2 output to the oscilloscope input.
3. Connect the P2 sync output to the oscilloscope. Synchronize the oscilloscope on the sync output.
4. Configure the P2 as follows:
   Output: On

Adjustment:
1. Adjust CAL:SETup 16 for a rise time reading of 2 ms, ±50 µs.

Pulse Leading Edge Adjustments

The pulse trailing edge adjustments assure that the trailing edge transition times of the generated pulses are within the specified limits. Note that each channel has its own trailing edge adjustments and therefore, after you do the adjustments on channel 1 continue with the same adjustments on channel 2. Use this procedure if you suspect that the trailing edge timing is not within range.

Setup 17

50 ns Trailing Edge

Equipment: oscilloscope

Preparation:

1. Configure the oscilloscope as follows:
   - Vertical: 1 V/div
   - Horizontal: 10 ns/div
   - Termination: 50 Ω
   - Measure: Fall time
2. Connect the P2 output to the oscilloscope input.
3. Connect the P2 sync output to the oscilloscope. Synchronize the oscilloscope on the sync output.
4. Configure the P2 as follows:
   - Output: On

Adjustment:

1. Adjust CAL:SETup 17 for a rise time reading of 50 ns, ±2 ns.

Setup 18

200 ns Trailing Edge

Equipment: oscilloscope

Preparation:

1. Configure the oscilloscope as follows:
   - Vertical: 1 V/div
   - Horizontal: 50 ns/div
   - Termination: 50 Ω
   - Measure: Fall time
2. Connect the P2 output to the oscilloscope input.
3. Connect the P2 sync output to the oscilloscope. Synchronize the oscilloscope on the sync output.
4. Configure the P2 as follows:
   - Output: On

Adjustment:

1. Adjust CAL:SETup 18 for a rise time reading of 200 ns, ±10 ns.

Setup 19

2 μs Trailing Edge

Equipment: oscilloscope

Preparation:

1. Configure the oscilloscope as follows:
   Vertical: 1 V/div
   Horizontal: 500 ns/div
   Termination: 50 Ω
   Measure: Fall time
2. Connect the P2 output to the oscilloscope input.
3. Connect the P2 sync output to the oscilloscope. Synchronize the oscilloscope on the sync output.
4. Configure the P2 as follows:
   Output: On

Adjustment:

1. Adjust CAL:SETup 19 for a rise time reading of 2 µs, ±50 ns.

Setup 20

20 µs Trailing Edge

Equipment: oscilloscope

Preparation:

1. Configure the oscilloscope as follows:
   Vertical: 1 V/div
   Horizontal: 5 µs /div
   Termination: 50 Ω
   Measure: Fall time
2. Connect the P2 output to the oscilloscope input.
3. Connect the P2 sync output to the oscilloscope. Synchronize the oscilloscope on the sync output.
4. Configure the P2 as follows:
   Output: On

Adjustment:

1. Adjust CAL:SETup 20 for a rise time reading of 20 µs, ±0.5 µs.

Setup 21

200 µs Trailing Edge

Equipment: oscilloscope

Preparation:

1. Configure the oscilloscope as follows:
   Vertical: 1 V/div
   Horizontal: 50 µs /div
   Termination: 50 Ω
   Measure: Fall time
2. Connect the P2 output to the oscilloscope input.
3. Connect the P2 sync output to the oscilloscope. Synchronize the oscilloscope on the sync output.
4. Configure the P2 as follows:
   Output: On

Adjustment:

1. Adjust CAL:SETup 21 for a rise time reading of 200 µs, ±5 µs.

Setup 22

2 ms Trailing Edge

Equipment: oscilloscope

Preparation:

1. Configure the oscilloscope as follows:
   Vertical: 1 V/div
   Horizontal: 0.5 ms/div
   Termination: 50 Ω
   Measure: Fall time
2. Connect the P2 output to the oscilloscope input.
3. Connect the P2 sync output to the oscilloscope. Synchronize the oscilloscope on the sync output.
4. Configure the P2 as follows:
   Output: On

Adjustment:

1. Adjust CAL:SETup 22 for a rise time reading of 2 ms, ±50 µs.

Pulse Vertical Balance Adjustments

The pulse vertical balance adjustments assure that the pulses are symmetrical about the 0 V center line. Note that each channel has its own adjustments and therefore, after you do the adjustments on channel 1 continue with the same adjustments on channel 2. Use this procedure if you suspect that the vertical balance is not within range.

Setup 23

Vertical Balance Step 1

Equipment: oscilloscope

Preparation:

1. Configure the oscilloscope as follows:
   Vertical: 1 V/div
   Horizontal: 0.2 ms/div
   Termination: 50 Ω
2. Connect the P2 output to the oscilloscope input.
3. Connect the P2 sync output to the oscilloscope. Synchronize the oscilloscope on the sync output.
4. Configure the P2 as follows:
   Output: On

Adjustment:

1. Adjust CAL:SETup 23 for a best vertical alignment about the vertical center line.

Setup 24

Vertical Balance Step 2

Equipment: oscilloscope

Preparation:

1. Configure the oscilloscope as follows:
   Vertical: 1 V/div
   Horizontal: 0.2 ms/div
   Termination: 50 Ω
2. Connect the P2 output to the oscilloscope input.
3. Connect the P2 sync output to the oscilloscope. Synchronize the oscilloscope on the sync output.
4. Configure the P2 as follows:
   Output: On

Adjustment:

1. Adjust CAL:SETup 24 for a best vertical alignment about the vertical center line.

Setup 25

Vertical Balance Step 3

Equipment: oscilloscope

Preparation:

1. Configure the oscilloscope as follows:
   Vertical: 1 V/div
   Horizontal: 0.2 ms/div
   Termination: 50 Ω
2. Connect the P2 output to the oscilloscope input.
3. Connect the P2 sync output to the oscilloscope. Synchronize the oscilloscope on the sync output.
4. Configure the P2 as follows:
   Output: On

Adjustment:

1. Adjust CAL:SETup 25 for a best vertical alignment about the vertical center line.

Setup 26

Vertical Balance Step 4

Equipment: oscilloscope

Preparation:

1. Configure the oscilloscope as follows:
   Vertical: 1 V/div
   Horizontal: 0.2 ms/div
   Termination: 50 Ω
2. Connect the P2 output to the oscilloscope input.
3. Connect the P2 sync output to the oscilloscope. Synchronize the oscilloscope on the sync output.
4. Configure the P2 as follows:
   Output: On

Adjustment:

1. Adjust CAL:SETup 26 for a best vertical alignment about the vertical center line.
2. Repeat setups 23 through 26 for best vertical alignments between the setups.

Baseline Offset Adjustments

The baseline offset adjustments assure that the pulse signal is symmetrical about the amplitude mid-range value for all amplitude ranges. Use this procedure if you suspect that there is a baseline accuracy issue.

Setup 27

Null Offset - Symmetrical Voltage Range

Equipment: DMM

Preparation:

1. Configure the DMM as follows:
   Function: DCV
   Range: 100 mV
2. Connect the P2 output to the DMM input. Use a 50 Ω feedthrough termination at the DMM input
3. Configure the P2 as follows:
   Output: On
   Amplitude: 10 mV

Adjustment:

1. Program CAL:SETup 27
2. Adjust RV5 (RV2 in channel 2) for a DMM reading of 0 V, ±20 mV.

Setup 28

Amplifier Offset - Positive Voltage Range

Equipment: DMM

Preparation:

1. Configure the DMM as follows:
   Function: DCV
   Range: 100 mV
2. Connect the P2 output to the DMM input. Use a 50 Ω feedthrough termination at the DMM input
3. Configure the P2 as follows:
   Output: On
   Amplitude Range: Positive
   Amplitude: 10 mV

Adjustment:

1. Adjust CAL:SETup 28 for a DMM reading of 0 V, ±20 mV.

Setup 29

Amplifier Offset - Negative Voltage Range

Equipment: DMM

Preparation:

1. Configure the DMM as follows:
   Function: DCV
   Range: 100 mV
2. Connect the P2 output to the DMM input. Use a 50 Ω feedthrough termination at the DMM input
3. Configure the P2 as follows:
   Output: On
   Amplitude Range: negative
   Amplitude: 10 mV

Adjustment:

1. Adjust CAL:SETup 29 for a DMM reading of 0 V, ±20 mV.

Setup 30

5 V Amplitude, 0 V Offset

Equipment: DMM

Preparation:

1. Configure the DMM as follows:
   Function: ACV
   Range: 10 V
2. Connect the P2 output to the DMM input. Use a 50 Ω feedthrough termination at the DMM input
3. Configure the P2 as follows:
   Output: On
   Period: 1 ms
   Amplitude: 5 V

Adjustment:

1. Adjust CAL:SETup 30 for a DMM reading of 0 V, ±20 mV.

Offset Adjustments

The offset adjustments assure that the DC offsets are within the specified range. Use this procedure if you suspect that the offset accuracy is an issue.

Setup 31

+1 V Offset

Equipment: DMM

Preparation:

1. Configure the DMM as follows:
   Function: DCV
   Range: 1 V

2. Connect the P2 output to the DMM input. Use a 50 Ω feedthrough termination at the DMM input.
3. Configure the P2 as follows:

Output:	On
Amplitude Range:	Positive
Period:	1 µs
Amplitude:	10 mV
Offset:	1 V

Adjustment:

1. Adjust CAL:SETup 31 for a DMM reading of 1 V, ±5 mV.

Setup 32

+2.5 V Offset

Equipment: DMM

Preparation:

1. Configure the DMM as follows:

Function:	DCV
Range:	10 V

2. Connect the P2 output to the DMM input. Use a 50 Ω feedthrough termination at the DMM input.
3. Configure the P2 as follows:

Output:	On
Amplitude Range:	Positive
Period:	1 µs
Amplitude:	10 mV
Offset:	2.5 V

Adjustment:

1. Adjust CAL:SETup 32 for a DMM reading of 2.5 V, ±10 mV.

Setup 33

+4 V Offset

Equipment: DMM

Preparation:

1. Configure the DMM as follows:

Function:	DCV
Range:	10 V

2. Connect the P2 output to the DMM input. Use a 50 Ω feedthrough termination at the DMM input.
3. Configure the P2 as follows:

Output:	On
Amplitude Range:	Positive
Period:	1 µs
Amplitude:	10 mV
Offset:	4 V

Adjustment:

1. Adjust CAL:SETup 33 for a DMM reading of 4 V, ±20 mV.

Setup 34

+5.5 V Offset

Equipment: DMM

Preparation:

1. Configure the DMM as follows:
   Function: DCV
   Range: 10 V
2. Connect the P2 output to the DMM input. Use a 50 Ω feedthrough termination at the DMM input.
3. Configure the P2 as follows:
   Output: On
   Amplitude Range: Positive
   Period: 1 µs
   Amplitude: 10 mV
   Offset: 5.5 V

Adjustment:

1. Adjust CAL:SETup 34 for a DMM reading of 5.5 V, ±25 mV.

Setup 35

+7 V Offset

Equipment: DMM

Preparation:

1. Configure the DMM as follows:
   Function: DCV
   Range: 10 V
2. Connect the P2 output to the DMM input. Use a 50 Ω feedthrough termination at the DMM input.
3. Configure the P2 as follows:
   Output: On
   Amplitude Range: Positive
   Amplitude: 10 mV
   Offset: 7 V

Adjustment:

1. Adjust CAL:SETup 34 for a DMM reading of 7 V, ±35 mV.

Setup 36

+8.5 V Offset

Equipment: DMM

Preparation:

1. Configure the DMM as follows:
   Function: DCV
   Range: 10 V
2. Connect the P2 output to the DMM input. Use a 50 Ω feedthrough termination at the DMM input.
3. Configure the P2 as follows:

Output:	On
Amplitude Range:	Positive
Period:	1 µs
Amplitude:	10 mV
Offset:	8.5 V

Adjustment:

1. Adjust CAL:SETup 34 for a DMM reading of 8.5 V, ±40 mV.

Setup 37

+9.5 V Offset

Equipment: DMM

Preparation:

1. Configure the DMM as follows:
   Function: DCV
   Range: 10 V
2. Connect the P2 output to the DMM input. Use a 50 Ω feedthrough termination at the DMM input.
3. Configure the P2 as follows:
   Output: On
   Amplitude Range: Positive
   Period: 1 µs
   Amplitude: 10 mV
   Offset: 9.5 V

Adjustment:

1. Adjust CAL:SETup 34 for a DMM reading of 9.5 V, ±40 mV.

Setup 38

-1 V Offset

Equipment: DMM

Preparation:

1. Configure the DMM as follows:
   Function: DCV
   Range: 1 V
2. Connect the P2 output to the DMM input. Use a 50 Ω feedthrough termination at the DMM input.
3. Configure the P2 as follows:
   Output: On
   Amplitude Range: Positive
   Period: 1 µs
   Amplitude: 10 mV
   Offset: -1 V

Adjustment:

1. Adjust CAL:SETup 31 for a DMM reading of -1 V, ±5 mV.

Setup 39

-2.5 V Offset

Equipment: DMM

Preparation:

1. Configure the DMM as follows:
   Function: DCV
   Range: 10 V
2. Connect the P2 output to the DMM input. Use a 50 Ω feedthrough termination at the DMM input.
3. Configure the P2 as follows:
   Output: On
   Amplitude Range: Positive
   Period: 1 µs
   Amplitude: 10 mV
   Offset: -2.5 V

Adjustment:

1. Adjust CAL:SETup 32 for a DMM reading of -2.5 V, ±10 mV.

Setup 40

-4 V Offset

Equipment: DMM

Preparation:

4. Configure the DMM as follows:
   Function: DCV
   Range: 10 V
5. Connect the P2 output to the DMM input. Use a 50 Ω feedthrough termination at the DMM input.
6. Configure the P2 as follows:
   Output: On
   Amplitude Range: Positive
   Period: 1 µs
   Amplitude: 10 mV
   Offset: -4 V

Adjustment:

2. Adjust CAL:SETup 33 for a DMM reading of -4 V, ±20 mV.

Setup 41

-5.5 V Offset

Equipment: DMM

Preparation:

1. Configure the DMM as follows:
   Function: DCV
   Range: 10 V
2. Connect the P2 output to the DMM input. Use a 50 Ω feedthrough termination at the DMM input.
3. Configure the P2 as follows:
   Output: On

Amplitude Range:	Positive
Period:	1 µs
Amplitude:	10 mV
Offset:	-5.5 V

Adjustment:

1. Adjust CAL:SETup 34 for a DMM reading of -5.5 V, ±25 mV.

Setup 42

-7 V Offset

Equipment: DMM

Preparation:

1. Configure the DMM as follows:

Function:	DCV
Range:	10 V

2. Connect the P2 output to the DMM input. Use a 50 Ω feedthrough termination at the DMM input.
3. Configure the P2 as follows:

Output:	On
Amplitude Range:	Positive
Period:	1 µs
Amplitude:	10 mV
Offset:	-7 V

Adjustment:

1. Adjust CAL:SETup 34 for a DMM reading of -7 V, ±35 mV.

Setup 43

-8.5 V Offset

Equipment: DMM

Preparation:

1. Configure the DMM as follows:

Function:	DCV
Range:	10 V

2. Connect the P2 output to the DMM input. Use a 50 Ω feedthrough termination at the DMM input.
3. Configure the P2 as follows:

Output:	On
Amplitude Range:	Positive
Period:	1 µs
Amplitude:	10 mV
Offset:	-8.5 V

Adjustment:

1. Adjust CAL:SETup 34 for a DMM reading of -8.5 V, ±40 mV.

Setup 44

-9.5 V Offset

Equipment: DMM

Preparation:

1. Configure the DMM as follows:
   Function: DCV
   Range: 10 V
2. Connect the P2 output to the DMM input. Use a 50 Ω feedthrough termination at the DMM input.
3. Configure the P2 as follows:
   Output: On
   Amplitude Range: Positive
   Period: 1 µs
   Amplitude: 10 mV
   Offset: -9.5 V

Adjustment:

1. Adjust CAL:SETup 34 for a DMM reading of -9.5 V, ±40 mV.

Amplitude Adjustments

The amplitude adjustments assure that the AC levels are within the specified range. Use this procedure if you suspect that the amplitude accuracy is an issue.

Setup 45

5 V Amplitude

Equipment: DMM

Preparation:

1. Configure the DMM as follows:
   Function: ACV
   Range: 10 V
2. Connect the P2 output to the DMM input. Use a 50 Ω feedthrough termination at the DMM input.
3. Configure the P2 as follows:
   Output: On
   Amplitude: 5 V
   Period: 1 ms

Adjustment:

1. Adjust CAL:SETup 45 for a DMM reading of 2.5 V, ±15 mV.

Setup 46

1.5 V Amplitude

Equipment: DMM

Preparation:

1. Configure the DMM as follows:
   Function: ACV
   Range: 1 V
2. Connect the P2 output to the DMM input. Use a 50 Ω feedthrough termination at the DMM input.

3. Configure the P2 as follows:
   Output: On
   Amplitude: 1.5 V
   Period: 1 ms

Adjustment:
1. Adjust CAL:SETup 46 for a DMM reading of 750 mV, ±5 mV.

Setup 47

0.5 V Amplitude

Equipment: DMM

Preparation:
1. Configure the DMM as follows:
   Function: ACV
   Range: 1 V
2. Connect the P2 output to the DMM input. Use a 50 Ω feedthrough termination at the DMM input.
3. Configure the P2 as follows:
   Output: On
   Amplitude: 0.5 V
   Period: 1 ms

Adjustment:
1. Adjust CAL:SETup 47 for a DMM reading of 250 mV, ±1.5 mV.

Setup 48

150 mV Amplitude

Equipment: DMM

Preparation:
1. Configure the DMM as follows:
   Function: ACV
   Range: 100 mV
2. Connect the P2 output to the DMM input. Use a 50 Ω feedthrough termination at the DMM input.
3. Configure the P2 as follows:
   Output: On
   Amplitude: 150 mV
   Period: 1 ms

Adjustment:
1. Adjust CAL:SETup 48 for a DMM reading of 75 mV, ±0.5 mV.

Setup 49

50 mV Amplitude

Equipment: DMM

Preparation:
1. Configure the DMM as follows:
   Function: ACV

Range: 100 mV

2. Connect the P2 output to the DMM input. Use a 50 Ω feedthrough termination at the DMM input
3. Configure the P2 as follows:
   Output: On
   Amplitude: 50 mV
   Period: 1 ms

Adjustment:

1. Adjust CAL:SETup 49 for a DMM reading of 25 mV, ±0.5 mV.

Setup 50

15 mV Amplitude

Equipment: DMM

Preparation:

1. Configure the DMM as follows:
   Function: ACV
   Range: 100 mV
2. Connect the P2 output to the DMM input. Use a 50 Ω feedthrough termination at the DMM input
3. Configure the P2 as follows:
   Output: On
   Amplitude: 15 mV
   Period: 1 ms

Adjustment:

1. Adjust CAL:SETup 50 for a DMM reading of 7.5 mV, ±0.5 mV.

Pulse Response Adjustments

The pulse response adjustments assure that the rise and fall times, as well as, the aberrations are within the specified limits. Use this procedure if you suspect that the pulse response is an issue. Note that setup 51 is marked as a factory adjustment and therefore, it is not normally required to be performed during normal calibration cycles except if the pulse response of the output stage has been degraded and does not meet the published specification limits, or after a repair has been executed on this very circuit.

(Setup 51)

5 V Amplitude

Equipment: oscilloscope

Preparation:

1. Connect the P2 output to the oscilloscope input. Use a 50 Ω feedthrough termination at the oscilloscope input.
2. Configure the P2 as follows:
   Output: On
   Amplitude: 6 V

Adjustment:

1. Program CAL:SETup 51
2. Adjust the trace on the oscilloscope to show 6 vertical divisions.
3. Adjust RV4 and RV7 (RV1 and RV8 in channel 2) for best pulse response (typically 8 ns transitions and 5% aberrations).

Adjustment Procedure – A3 Module

Use the following procedures to calibrate the Model A3 module. The following paragraphs show how to set up the instrument for calibration and what the acceptable calibration limits are.

Amplifier Gain Adjustments

The gain adjustments ensure that amplifier gain within the specified range. Use this procedure if you suspect that the gain is an issue.

Gain Adjustments

Equipment: DMM

Preparation:

1. Configure the DMM as follows:
   Function: ACV
2. Connect the 3172-W6 output to DMM input
3. Configure the 3172-W6 as follows:
   Wave Shape: Sine
   Frequency: 10 kHz
   Amplitude: 10Vpp
   Output: On
4. Adjust 3172-W6 amplitude to DMM reading is 3.535 V.
5. Connect the 3172-W6 output to the A3 input.
6. Connect the A3 output to the DMM input.

Adjustment

1. Adjust RV1 till DMM reading is within 42.42V, ± 500mV

Pulse Response Adjustments

The pulse response adjustments ensure that rise and fall times as well as aberrations are within the specified range. Use this procedure if you suspect that the pulse response is an issue.

Pulse Adjustments

Equipment: Oscilloscope

Preparation:

1. Configure the 3172-W6 as follows:
   Function: Standard
   Waveform: Square
   Frequency: 1MHz
   Amplitude: 8Vpp
   Output: On
2. Connect the 3172-W6 output to the A3 Input.
3. Connect the A3 output to the oscilloscope via 20dB feed through attenuator.

Adjustment:

1. Adjust C3 for best pulse response (rise/fall time of <1.5µs, 15% aberrations).

Updating the Firmware

WARNING

Only qualified persons may perform firmware updates. DO NOT attempt to perform this operation unless you are trained, as you may otherwise render the instrument inoperable. Always verify with customer service that you have the latest firmware file before starting an update.

Updating Message-based Firmware

Before you attempt to update the firmware, check the revision level of the product. Each firmware update was done for a reason and therefore, if you want to update the firmware to address a specific problem in your system, check the readme file that is associated with the update to see if an update will solve your problem. There are a number of ways to check the revision level of your firmware:

1. Use a SCPI command from an external utility. You can read the firmware version by sending the following query:

   `SYST:INFO:FIRM:VERS?`

   The response is a string showing the firmware version, e.g., 3.07.

2. Using ArbConnection, select the General/Filters panel from the System tab and click on the Firmware Version button.

The computer that will be used for performing the update must have the 3172 VXI*plug&play* driver installed. If it does not, then you may download the current version from the Astronics Test Systems software download web site (search for 3172 *VXIplug&play*).

Use the following procedure to upgrade the firmware once you determine that a firmware upgrade will improve the performance of your instrument.

1. Install the 3172 in a VXIbus chassis and apply power to the chassis.
2. Run the VXI resource manager (Resman.exe).
3. Copy the new 3172 object file to a folder on the hard disk. The object file has a filename containing the model number and version number. For example, the file "ri3172-v3.21.bin" is for Model 3172, version 3.21.
4. Launch "updater.exe". To locate it, navigate to the installation path of the 3172 *VXIplug&play* driver. The path is typically:

   C:\Program Files\IVI Foundation\VISA\WinNT\ri3172

   In this folder is the "firmware updater" folder, containing the file "updater.exe". Double-click "updater.exe" to launch the updater.
5. The updater window will appear as shown in Figure 7-18. In the Resource Name pull-down menu, select the logical address of the module to be updated. In the example shown in Figure 7-18, the item "VXIO::2::INSTR" refers to logical address 2.

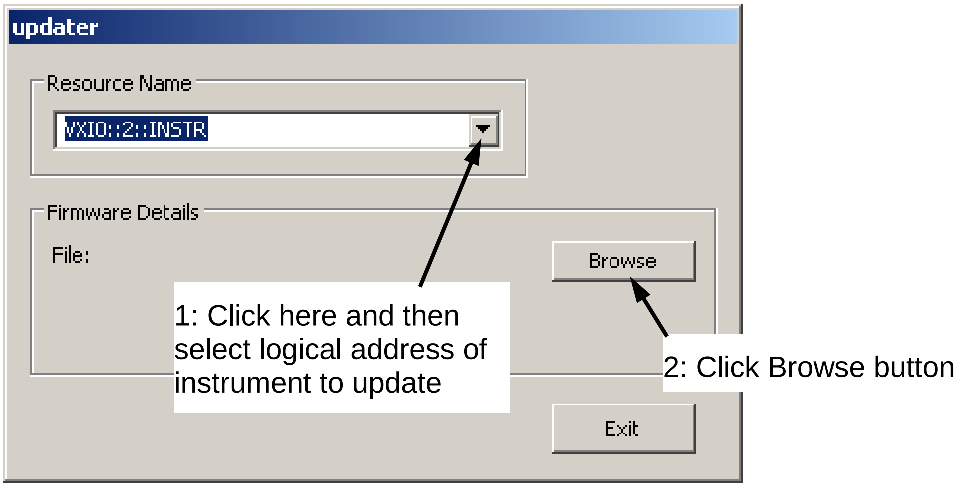

Figure 7-18, Updater Window

6. Click the Browse button. A file selection window will open.
7. Navigate to the folder where you placed the 3172 object file in step 3. Select the file (for example "ri3172 v3.07.bin") and then click Open.
8. The updater window will then include the Update button as shown in Figure 7-19.

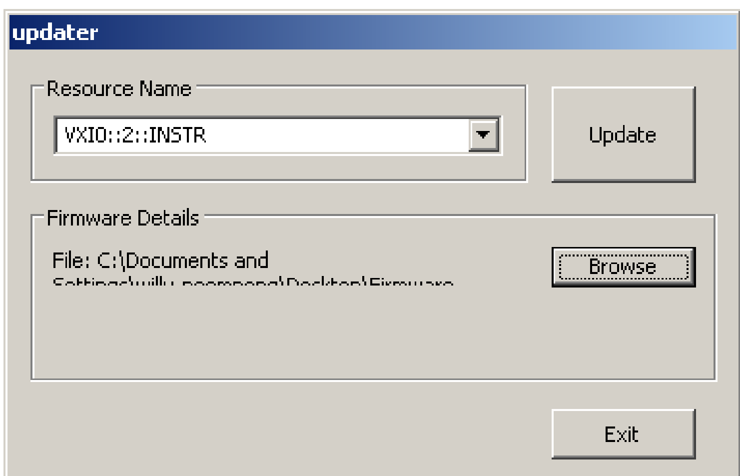

Figure 7-19, Updater Window with Update Button

CAUTION
Before clicking on the Update button in the next step below, be ready to observe the Fail LED on the 3172, and follow the instructions carefully.

9. Click the Update button and then immediately watch the Fail LED on the 3172 front panel. Wait for the LED to come on before proceeding. This will typically take between 5 and 20 seconds. If the Fail LED does not come on within two minutes, turn off power to the VXIbus chassis and contact the factory for further instructions.
10. After the Fail LED comes on, wait until it turns off or until at least one minute elapses.
11. By this time, a download success message will have appeared (Figure 7-20). Click the OK button. Then click the Exit button in the updater window.

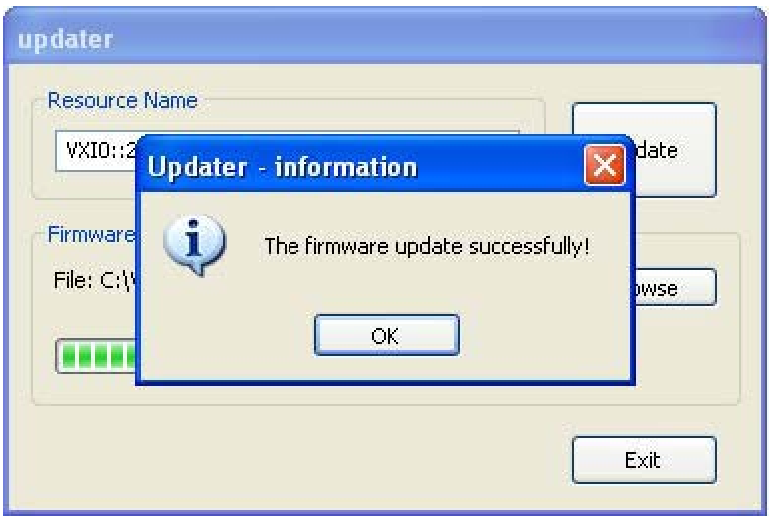

Figure 7-20, Successful Firmware Update

12. Turn off power to the VXI chassis. Wait at least two seconds, and then turn it back on.
13. Run the Resource Manager (Resman).
14. Send a *IDN? command to the 3172. The response will be a text string stating the model number and firmware version, as in this example:

    `Racal Instruments,3172,0,3.07\r\n`

    In this example, the firmware version is 3.07.
15. Verify that the firmware version is the one you loaded.

The updated instrument is ready to use.

This page was intentionally left blank.

Appendix A
3172 Module Specifications

3172 VXI Module Specifications

Backplane Multiple Instrument Synchronization

Description	Multiple instruments can be daisy-chained and synchronized to provide multi-channel systems. Phase (leading edge) offset between master and slave units is programmable.
Initial Skew	<±75 ns + 4 sample clock cycle
Synchronized Waveforms	Standard, Arbitrary and Sequenced using the automatic sequence advance mode only
Synchronized Run Modes	Continuous, Triggered, Gated and Counted Burst
Trigger Delay	200 ns to 20 s, separately programmable for each synchronized instrument
Trigger Delay Resolution	20 ns
Synchronization Frequency Range	
Standard Waveforms	1.5 kHz to 30 MHz
Arbitrary and Sequenced Waveforms	2.5 MSa/s to 150 MSa/s, limited by a specific VXI backplane ability to transmit high frequency signals.

Leading Edge Offset

Description	Leading edge offset is programmable for master and slave units. Operates in conjunction with the continuous run mode only
Offset Range	<100 ns to 20 s
Resolution and Accuracy	20 ns

VXIbus General Information

Module Form	Single slot, C-size module
Protocol	A16/A24/A32/D16 Slave, Message or Register-Based
VXIbus Revision	3.0

SCPI Revision	1993.0
SCPI Emulation	3171
Logical Address Settings	1 - 255, configured via DIP switches
Shared Memory (Write only)	D16/A24/A32

General

Power Requirements	35W maximum
EMC Certification	CE marked
Operating temperature	0 °C - 55 °C
Storage Temperature	-40°C - 70 °C
Safety	Designed to meet IEC 1010-1, UL 3111-1, CSA 22.2 #1010
Workmanship Standards	Conform to IPC-A-610D

W6 (Arbitrary Waveform Generator) Specifications

Run Modes

Description	Define how waveforms start and stop. Run modes description applies to all waveform types and functions, except where noted otherwise.
Continuous	Continuously free-run output of a waveform. Waveform generation can be enabled and disabled from a remote interface only.
Triggered	Upon trigger, outputs one or more waveform cycles. Burst counter controls the number of waveform cycles. Last cycle always completed. Trigger can be internal or external.
Gated	Transition or level enables or disables generator output. Last cycle always completed.

Frequency Accuracy

10 MHz Reference Source	Internal, External, VXI backplane CLK10
Internal	≥ 0.0001% (1 ppm TCXO) initial tolerance from 19°C to 29°C; 1ppm/°C below 19°C and above 29°C; 1 ppm/year aging rate
External	
Connector	Front panel Combo D-sub 24W7 – A4
Impedance and Level	10 kΩ ±5%, TTL, 50% ±2% duty cycle, or 50 Ω ±5%, 0 dBm, manually selectable using internal jumpers
External Sample Clock Source	External, ECLTRG0
Connector	Front panel Combo D-sub 24W7 – A8
Frequency	From DC to 200 MHz, external; DC to 66MHz,

	ECLTRG0
Impedance and Level	50 Ω ±5%, PECL or NECL

Output Characteristics

Main Output

Channels	1
Connector	Front panel Combo D-sub 24W7 – A7
Impedance	Selectable <2 Ω, 50 Ω, or 93 Ω, nominal
Protection	Short Circuit to Case Ground, 400 mA current limit
Standby	Output ON or OFF (Output Disconnected)
Amplitude	
Window (Zout = <2 Ω)	NEG mode: 0 to -16 V POS mode: 0 to 20 V SYMM mode: -11 V to 11 V
Window (Zout = 50 Ω)	NEG mode: 0 to -8 V POS mode:0 to 10 V SYMM mode: -5.5 V to 5.5 V
Level (Zout = <2 Ω)	SYMM mode: 5 mV to 22 Vp-p
Level (Zout = 50 Ω)	SYMM mode: 2.5 mV to 11 Vp-p
Resolution	12-bits
Accuracy (measured at 1 kHz into 50 Ω)	
1 Vp-p to 11 Vp-p	±(1% of setting + 50 mV)
100 mVp-p to 1 Vp-p	±(1% of setting + 20 mV)
10 mVp-p to 100 mVp-p	±(1% of setting + 5 mV)
Accuracy (measured at 1 kHz into 93 Ω)	
1 Vp-p to 11 Vp-p	±(2% of setting + 50 mV)
100 mVp-p to 1 Vp-p	±(2% of setting + 20 mV)
10 mVp-p to 100 mVp-p	±(2% of setting + 5 mV)
Accuracy (measured at 1 kHz into >1M Ω)	
1 Vp-p to 11 Vp-p	±(2.5% of setting + 100 mV)
100 mVp-p to 1 Vp-p	±(2.5% of setting + 40 mV)
10 mVp-p to 100 mVp-p	±(2.5% of setting + 10 mV)
DC Offset Range	NEG mode: 0 to -16 V POS mode: 0 to 20 V SYMM mode: 0 V to ±11 V
Resolution	1 mV
Accuracy, into 50Ω	±(1% ± 1% from Amplitude ±15 mV)
Accuracy, into 93Ω	±(2% ± 1% from Amplitude ±15 mV)
Accuracy, into ≥1MΩ	±(2.5% ± 2% from Amplitude ±30 mV)

Rise/Fall Time (10%-90%)	11 ns
Aberration (Zout = 50 Ω)	< 6.5%

Sync Output

Connector	Front panel Combo D-sub 24W7 – A5; TTLTRG0-7
Level	TTL
Sync Type	Pulse with Arbitrary and Standard Waves; LCOM in Sequence and Burst Modes (including Burst Modulation); Marker with Modulation Mode only, programmable position
Width	4 to 64 waveform points
Position	Point 0 to maximum segment size, programmable with 4-point resolution

Filters

Description	Filters can be switched in and out freely except in standard waveform shape where the filters are automatically used by the instrument to reconstruct the sine shape.
Type	2 MHz, Bessel; 25 MHz Bessel; 60 MHz Elliptic; 120 MHz Elliptic

Trigger Input Characteristics

Sources

Internal	Asynchronous timer generates triggers repeatedly
Timer Range	1 μs to 20 s
External	External trigger input is enabled
Input	Front panel Combo D-sub 24W7 – A3
Impedance	10 kΩ
Range	±10 V
Resolution	10 mV
Sensitivity	200 mVrms
Damage Level	30 Vrms
Frequency Range	DC to 5 MHz
Slope	Positive/Negative transitions, selectable
Minimum Pulse Width	≥ 10 ns
BUS	Trigger commands from a remote controller only
VXI Backplane	TTLTRG0-7 inputs or outputs, programmable
System Delay (Trigger input to waveform output)	6 sample clock cycles+150 ns

Trigger Delay (Trigger input to waveform output)	0; 100 ns to 20 s
Resolution	20 ns
Error	6 sample clock cycles + 250 ns + 5% of setting
Re-trigger Delay (Waveform end to waveform restart)	100 ns to 20 s
Resolution	20 ns
Error	3 sample clock cycles + 20 ns + 5% of setting
Trigger Jitter	±1 sample clock period
Burst Counter	1 to 1,000,000, programmable

PLL Characteristics

Description	Automatically locks 3172 output to external signal applied to trigger port
PLL Input Characteristics	Same as trigger input
External Lock Frequency Range	Standard Waveforms: 500 Hz to 10 MHz
	Arbitrary Waveforms: 500 Hz to 100M/(points per cycle)
Phase Control	Coarse: ± 180º
	Fine: ± 36º with 0.01 º resolution
Fine Phase Control Accuracy	10%

PM Characteristics

Description	External signal offsets phase. Available in PLL mode when unit is locked to an external signal.
PM Input Bandwidth	100 Hz-10 kHz
External Lock Frequency Range	Standard Waveforms: 500 Hz to 10 MHz
	Arbitrary Waveforms: 500 Hz to 100 M/(pts per cycle)
PM Input	
Connector	Front panel Combo D-sub 24W7 – A2
Impedance	100 kΩ ±5%
Sensitivity	20º / V
Range	±130º
PM Accuracy	±30%

External AM Characteristics

Description	External signal amplitude modulates the carrier signal
Input	Front panel Combo D-sub 24W7 – A1
Impedance	10 kΩ
Coupling	DC
Bandwidth	1 MHz
Range	W2: 0 V to ±5 V controls modulation depth from 0% to 100%, respectively
	W6: 0 V to ±10 V controls modulation according to Vout – Vprog +2 *AM IN

Function Generator Characteristics

Description	One may select from a list of a built-in library of standard waveforms. The waveforms are computed every time a waveform is selected. The integrity of the waveform and its upper frequency limit depend on the programmed frequency value and the number of waveform points that are used for computing one cycle interval
Standard Functions	Sine, Triangle, Square, Pulse, Ramp, Sinc pulse Gaussian pulse, Exponential decay/rise pulse, DC
Frequency Range	
Sine, Square	100 μHz to 30 MHz
All other waveforms	100 μHz to 10 MHz
Frequency Resolution	11 digits
Accuracy & Stability	Same as frequency reference

Sine

Start Phase Range	0-359.95°
Start Phase Resolution	0.05°
Power Range (Sine Raised to a power)	1-9

Sine Wave Performance (standard and arbitrary waveforms)

THD	0.2% to 100 kHz, STD and CW
Harmonics and Spurious at less than 5 Vp-p	29 dBc, <25 MHz: 23 dBc <30 MHz
	44 dBc, <10 MHz
Harmonics & Spurious at less than 10 Vp-p	23 dBc, <30 MHz
	35 dBc, <10 MHz
Flatness at less than 5 Vp-p	10% to 10 MHz
	30% to 30 MHz

Triangle

Start Phase Range	0-359.9°
Start Phase Resolution	0.05°
Power Range (Triangle Raised to a power)	1-10

Square

Duty Cycle Range	0% to 99.9%

Pulse

Delay, Rise/Fall Time, High Time Ranges	0%-99.99% of period (each independently)

Ramp

Delay, Rise/Fall Time, High Time Ranges	0%-99.9% of period (each independently)

Gaussian Pulse

Time Constant Range	1-200

Sinc Pulse

"Zero Crossings" Range	4-100

Exponential Pulse

Time Constant Range	-200 to 200

DC Output Function

Range	±100% of amplitude setting

Arbitrary Waveform Generator Characteristics

Description	Arbitrary Waveforms are created on a remote computer and downloaded to the instrument through one of the available remote interfaces. The frequency of the waveform is calculated from its programmed sample clock value and the number of waveform points that were used for creating the waveform
Sample Clock Range	100 mSa/s to 200 MSa/s
Resolution	11 digits
Accuracy and Stability	Same as reference
Native Vertical Resolution	16 bits (65,536 amplitude increments)
3171 Emulation Vertical Resolution	12 bits (automatically shifts legacy data to 16 bit format)
Waveform Segmentation	Permits division of the waveform memory into smaller segments.
Number of Memory Segments	1 to 16k
Waveform Segments, size and resolution	2 point size increments from 10 to 1M points
Custom Waveform Creation Software	ArbConnection software allows instrument control and creation of custom waveforms

Sequenced Waveforms Generator Characteristics

Description	Segments may be linked and repeated in a user-selectable order. Segments are advanced using either a command or a trigger
Sample Clock Range	100 mSa/s to 200 MSa/s
Resolution	11 digits
Accuracy and Stability	Same as reference
Advance Modes	
Automatic Sequence Advance	No trigger required to step from one segment to the next. Sequence is repeated continuously per a pre-programmed sequence table.
Stepped Sequence Advance	Current segment is sampled continuously until a trigger advances the sequence to the next programmed segment and sample clock rate.
Single Sequence Advance	Current segment is sampled the specified number of repetitions and then idles at the end of the segment. Next trigger samples the next segment the specified repeat count, and so on.
Mixed Sequence Advance	Each step of a sequence can be programmed to advance either a) automatically (Automatic Sequence Advance), or b) with a trigger (Stepped Sequence Advance)
Sequencer Steps	1 to 4096
Segment Loops	1 to 1Meg
Minimum Segment Duration	500 ns
Minimum Segment Size in a Sequence	10 points
Custom Sequence Creation Software	ArbConnection software allows instrument control and creation of custom sequences. Sequences are built as tables and downloaded to the instrument

Internally Modulated Waveform Generator Characteristics

General

Description	Using this mode of operation, one may select from a list of built-in modulation schemes.
Carrier Waveform (CW)	Sinewave
Modulation Source	Internal only
Run Modes	Off (outputs CW), Continuous, Triggered, Delayed Trigger, Re-trigger, Burst and Gated
Interrupted Modulation Carrier Idle Mode	On or Off, programmable
Run Mode Advance Source	Front panel TRIG IN, TTLTRG(0-7), Internal trigger, Software commands
Trigger Delay (Trigger input to	

modulation output)	0; < 100 ns to 20 s
Resolution	20 ns
Error	6 sample clock cycles + 250 ns + 5% of settings
Re-trigger Delay (Modulation end to modulation restart)	< 100 ns to 20 s
Resolution	20 ns
Error	3 sample clock cycles + 20 ns + 5% of settings
Trigger Parameters	All trigger parameters such as level, slope, jitter, etc. apply

Marker Output

Description	Marks the crossing of a specific frequency and step setting. The marker pulse is generated through the SYNC connector. Marker placement is available for all modulation modes except external AM
Output	Front panel Combo D-sub 24W7 – A5, TTLTRG(0-7)
Level	TTL
Marker Type	Single pulse at the specified frequency
Position	Programmable for a specific frequency setting

Sweep

Swept Waveform	Sine, square or triangle
Sweep Step	Linear or log
Sweep Direction	Up or Down
Sweep Range	10 Hz to 30 MHz
Sweep Time	1.4 μs to 40 s

FM

Modulated Waveform	Sine wave
Modulating Waveforms	Sine, square, triangle, Ramp
Carrier Frequency Range	10 Hz to 30 MHz
Modulating Frequency Range	10 mHz to 350 kHz
Peak Deviation	Up to 15 MHz

ARBITRARY FM

Description	Operated from an external utility only such as ArbConnection. The modulating waveform can be designed as an arbitrary waveform
Modulated Waveform	Sine wave
Carrier Frequency Range	10 Hz to 30 MHz
Modulating Waveform	Arbitrary waveform
Modulating Waveform Sampling Clock	1 Sa/s to 2.5 MSa/s
Number of frequencies	2 to 10,000

AM (internal)

Modulated Waveform	Sine wave
Carrier Frequency Range	10 Hz to 30 MHz
Envelope Waveform	Sine, square, triangle, Ramp
Envelope Frequency	10 mHz to 100 kHz
Modulation Depth	0% to 200%

FREQUENCY HOPS

Hopped Waveform	Sine wave
Hop Frequency Range	10 Hz to 30 MHz
Resolution	11 digits
Hop Table Size	2 to 1000
Dwell Time Mode	Fixed or Programmable for each step
Dwell Time	200 ns to 20 s
Dwell Time Resolution	20 ns

AMPLITUDE HOPS

Hopped Waveform	Sine wave
Frequency Range	10 Hz to 30 MHz
Resolution	11 digits
Hop Amplitude Range	NEG mode: -2.5 to -16 Vp-p POS mode: 2.5 to 20 Vp-p SYMM mode: 2.5 V to 22 Vp-p
Resolution	FSR of maximum amplitude/4096
Hop Table Size	2 to 5000
Dwell Time Mode	Fixed or Variable for each step
Dwell Time	200 ns to 20 s
Dwell Time Resolution	20 ns

FSK

Shifted Waveform	Sine wave
Carrier/Shifted Frequency Range	10 Hz to 30 MHz
Baud Range	1 bit/sec to 10 Mbits/sec
FSK Data Bits Length	2 to 4000

PSK

Shifted Waveform	Sine wave
Carrier Frequency Range	10 Hz to 30 MHz
Phase Shift Range	0° to 359.99°
Baud Range	1 bits/sec to 10 Mbits/sec
PSK Data Bits Length	2 to 4000

ASK

Shifted Waveform	Sine wave
Carrier Frequency Range	10 Hz to 30 MHz
Amplitude Shift Range	0 V to 20 Vp-p
Resolution	Maximum amplitude/4096
Baud Range	1 bit/sec to 10 Mbits/sec
ASK Data Bits Length	2 to 4000

Counter/Timer Characteristics

Operation	The 3172 has a special mode where the instrument type is transformed to operate as a counter/timer. When this mode is selected, the operation of the arbitrary waveform and its outputs are disabled
Measurement Functions	Frequency, Period, Period Averaged, Pulse Width and Totalize

Frequency, Period Averaged

Frequency Range	20 Hz to 100 MHz (typically >120 MHz)
Period Averaged Range	10 ns to 50 ms
Resolution	7 digits in one second of gate time, reduced proportionally with lower gate times

Period, Pulse Width

Range	500 ns to 50 ms
Resolution	100 ns

Totalize

Frequency Range	20 Hz to 100 MHz
Accumulation Range	1 to 10^{12}-1

General

Input	Front panel TRIG IN, BNC connector
Trigger Level Range	±10 V
Sensitivity	500 mVp-p
Damage Level	±12 V
Minimum Pulse Width	≥10 ns
Slope	Positive/Negative transitions, selectable
Gate Time	100 µs to 1 s
Reading Modes	
Repetitive	Continuous measurements are executed when signal is present at the input
Hold	Single measurement is executed upon command

Gated	Active in Gated Totalize mode only
Time Base	
Type	TCXO
Temperature Stability	1 ppm, 0°C - 40°C
Long Term Stability	1 ppm, 1 year

P2 (Pulse Generator) Specifications

Pulse Generator Characteristics

Channel Dependency	Operation of channels is completely independent of each other but both share the 10 MHz reference source and VXI backplane triggers.
Operating Modes	Single pulse, delayed pulse, double pulse
Pulse Width Control	Programmable width, hold duty cycle, external width
Pulse Polarity	Normal, inverted or complemented
Pulse transitions	Fixed, or linear and programmable transitions

Run Modes

Description	Define how waveforms start and stop. Run modes description applies to all waveform types and functions, except where noted otherwise.
Continuous	Continuously free-run output of a waveform. Waveform generation can be enabled and disabled from a remote interface only.
Triggered	Upon trigger, outputs one or more waveform cycles. Burst counter controls the number of waveform cycles. Last cycle always completed.
Gated	Transition or level enables or disables generator output. Last cycle always completed.

Time Accuracy

10 MHz Reference Source	Internal, common for both channels, or VXI backplane CLK10
Accuracy and Stability	≥ 0.0001% (1 ppm TCXO) initial tolerance from 19°C to 29°C; 1ppm/°C below 19°C and above 29°C; 1 ppm/year aging rate

Output Characteristics

Main Output

Channels	2
Connector	Front panel Combo D-sub 5W5 – A1 and A5
Impedance	Selectable <2 Ω, 50 Ω, or 93 Ω, nominal

Protection	Short Circuit to Case Ground, 400 mA current limit
Standby	Output ON or OFF (Output Disconnected)
Amplitude	
Window (Zout = <2 Ω)	NEG mode: 0 to -16 V POS mode: 0 to 20 V SYMM mode: -11 V to 11 V
Window (Zout = 50 Ω)	NEG mode: 0 to -8 V POS mode: 0 to 10 V SYMM mode: -5.5 V to 5.5 V
Level (Zout = <2 Ω)	SYMM mode: 5 mV to 22 Vp-p
Level (Zout = 50 Ω)	SYMM mode: 2.5 mV to 11 Vp-p
Resolution	12-bits
Accuracy (measured at 1 kHz into 50 Ω)	
1 Vp-p to 11 Vp-p	±(1% of setting + 50 mV)
100 mVp-p to 1 Vp-p	±(1% of setting + 20 mV)
10 mVp-p to 100 mVp-p	±(1% of setting + 5 mV)
DC Offset Range	NEG mode: 0 to -16 V POS mode: 0 to 20 V SYMM mode: 0 V to ±11 V
Resolution	1 mV
Accuracy	±(1% ± 1% from Amplitude ±15 mV)
Rise/Fall Time (10%-90%)	11 ns
Aberration (Zout = 50 Ω)	< 6.5%

Sync Output

Connector	Front panel Combo D-sub 5W5 – A4; TTLTRG0-7
Level	TTL
Continuous Run Mode	
Sync Type	Pulse, fixed position
Width	10 to 20 ns, typical
Interrupted Run Modes (triggered, gated)	
Sync Type	Pulse, same width as trigger signal

Trigger Input Characteristics

Sources

External	External trigger input is enabled
Connector	Front panel Combo D-sub 5W5 – A3; TTLTRG0-7
Level	TTL, pulled up to +5 V through a 4.7 kΩ resistor
Damage Level	10 Vrms

Frequency Range	DC to 3 MHz
Slope	Positive/Negative transitions, selectable
Minimum Pulse Width	≥ 10 ns
BUS	Trigger commands from a remote controller only
VXI Backplane	TTLTRG0-7 inputs or outputs, programmable
System Delay (Trigger input to waveform output)	100 ns
Trigger Delay (Trigger input to waveform output)	0; 100 ns to 7 s
Resolution	20 ns
Trigger Jitter	±1 sample clock period
Burst Counter	1 to 65,536, programmable

Gate Input

Connector	Front panel Combo D-sub 5W5 – A2; TTLTRG0-7
Level	TTL, pulled up to +5 V through a 4.7 kΩ resistor

Controlled Pulse Parameters

Period

Range	20 ns to 5 s
Resolution	4 digits, continuous; 3 digits, gated and burst
Accuracy	0.01% (100 ppm), continuous; ±3%, all other run modes
Jitter	<(10 ppm + 20 ps) rms, Continuous; <(100 ppm + 20 ps) rms, Gate, Burst

Pulse Width, Double Pulse

Range	8 ns to 2.5 s
Delay	0 to 2.5 s
Resolution	5 digits limited by 10 ps
Accuracy	±(0.1% + 4 ns)
Jitter	<(100 ppm + 15 ps) rms

Linear Transitions

Description	Adjustable rising or falling edge, measured from 10% to 90% of amplitude
Range	5 ns to 5 ms in 6 overlapping ranges
In-range Span	20:1
Resolution	4 digits

Linearity	±3% of setting, above 100 ns
Accuracy	±(10% of setting + 2 ns)

Hold Duty Cycle Mode

Description	Output duty cycle remains constant regardless of period setting
Range	1% to 99%
Resolution	0.001%
Accuracy	±(3% of setting + 500 ps)

External Width

Description	The pulse shape can be recovered whilst the period and width of an external input signal are maintained
Input	Front panel EXT WID, SMB connector
Level and Slope	Trigger parameters apply

A3 Specifications

Input Characteristics

General

Connector	BNC
Impedance	50Ω, DC coupled

Amplifier Mode

Damage Level	50Vp-p
Frequency Range	DC to 100KHz

Bypass Mode

Signal source	Input connected directly to Output

Output Characteristics

General

Connector	BNC
Impedance	0.1 Ω, DC coupled
Protection	Short-circuit, 10 seconds
Gain	x12, fixed
Polarity	Normal
Amplitude	0 to 122 Vp-p (±61V)
Output Current	500 mA Max

Square Wave Characteristics

Transition Time	<1.5μs (typical)
Aberrations	<15% (typical)

Sine Wave Characteristics

Bandwidth (-3dB)	100 KHz, at 122 Vpp
Accuracy at Sine wave (1 KHz)	±3% of full-scale
THD	<0.1%, 10 Hz to 10 KHz
	<1.2%, 10 KHz to 100 KHz

General

Power Requirements	3.3 V, 5 V, ±24 V, ±12 V, all from VXIbus interface

Environmental

Operating Temperature	0 °C - 50 °C, RH 80% (non-condensing)
Storage Temperature	-30 °C - 80 °C