User Manual
8025/8026
100 MS/s SINGLE / DUAL
ARBITRARY WAVEFORM GENERATOR
Publication No. 030817

PUBLICATION DATE: August 17, 2003

REVISION: 1.2

WARRANTY STATEMENT

Products sold by Tabor Electronics are warranted to be free from defects in workmanship or materials. Tabor Electronics will, at its option, either repair or replace any hardware products, which prove to be defective during the warranty period. You are a valued customer. Our mission is to make any necessary repairs in a reliable and timely manner.

Duration of Warranty

The warranty period for this Tabor Electronics hardware is three years, except software and firmware products designed for use with Tabor Electronics Hardware is warranted not to fail to execute its programming instructions due to defect in materials or workmanship for a period of ninety (90) days from the date of delivery to the initial end user.

Return of Product

Authorization is required from Tabor Electronics before you send your product for service or calibration. Call your nearest Tabor Electronics support facility. A list is located on the last page of this manual. If you are unsure where to call, contact Tabor Electronics Customer Support Department in Tel Hanan, Israel at (972) 4 821 3393 or via fax at (972) 4 821 3388.

Limitation of Warranty

Tabor Electronics shall be released from all obligations under this warranty in the event repairs or modifications are made by persons other than authorized Tabor Electronics service personnel or without the written consent of Tabor Electronics.

Tabor Electronics expressly disclaims any liability to its customers, dealers and representatives and to users of its product, and to any other person or persons, for special or consequential damages of any kind and from any cause whatsoever arising out of or in any way connected with the manufacture, sale, handling, repair, maintenance, replacement or use of said products.

Representations and warranties made by any person including dealers and representatives of Tabor Electronics, which are inconsistent or in conflict with the terms of this warranty (including but not limited to the limitations of the liability of Tabor Electronics as set forth above), shall not be binding upon Tabor Electronics unless reduced to writing and approved by an officer of Tabor Electronics

Except as stated above, Tabor Electronics makes no warranty, express or implied (either in fact or by operation of law), statutory or otherwise; and except to the extent stated above, Tabor Electronics shall have no liability under any warranty, express or implied (either in fact or by operation of law), statutory or otherwise.

PROPRIETARY NOTICE

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" circuits points.

Before operation this instrument:

1. Ensure the instrument is configured to operate on the voltage at the power source. See Installation Section.

2. Ensure the proper fuse is in place for the power source to operate.

3. 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

Chapter 5
ArbExplorer

<div align="right">

Chapter 1

GETTING STARTED

</div>

What's In This Chapter

This chapter contains a general description of the Model 8025/8026 Arbitrary Waveform Generator and an overall functional description of the instrument. It lists and describes various options available for this model. It also describes the front panel connectors and indicators.

This manual provides description of all features and options available with the instruments; however, some items described in the following paragraphs and chapters may not be installed in your instrument. If you purchased Model 8025, ignore all references in this manual to Channel 2.

Introduction

Model 8025 is a single-channel Arbitrary Waveform Generator. Model 8026 is a dual-channel, Arbitrary Waveform Generator. It is a high performance waveform generator that combines two separate and powerful channels in one small package. Supplied free with the instrument is ArbExplorer software, which is used for controlling the 8026, generating, editing and downloading waveforms from a remote computer. The following highlights the 8026 and ArbExplorer features.

8026 Feature Highlights

- Dual output configuration with Independent waveform control

- Tight phase offset control between channels (1 point resolution)

- 14-bit vertical resolution

- Nearly 19-bit offset resolution

- 4 Meg memory depth for each channel (optional)

- Ultra fast waveform downloads using DMA

- 100 MS/s sample clock frequency

- 100 MHz sinewave output

- 1 ppm clock stability

- Extremely low phase noise carrier

- External amplitude modulation

- Frequency agility: FSK, ramped FSK, sweep, FM

- Trigger start phase control and breakpoints

- Built-in standard waveforms

- Separate sequence generators for each channel

- Multiple instrument synchronization with tight phase control

- GPIB and RS232 links

Figure 1-1, Model 8026

ArbExplorer Feature Highlights

- Three powerful tools in one software package: Instrument control panel, Waveform composer and FM signal composer

- Detailed virtual front panels control all 8026 functions and modes

- Wave composer generates, edits and downloads complex waveforms

- FM wave composer generates and downloads complex modulating signals

- Automatic detection of active instruments

- Equation editor generates waveforms from equations

- SCPI command and response editor simulates ATE operation

- Translates waveform coordinates from ASCII and other formats

- Simplifies generation of complex sequences
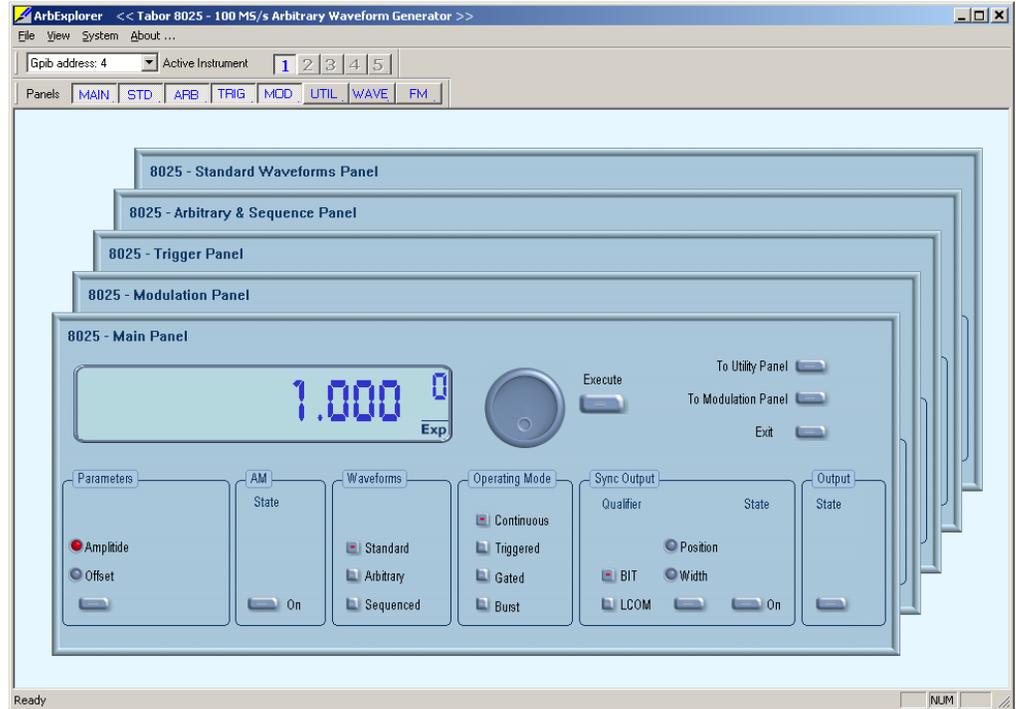
Figure 1-2, ArbExplorer - Control Panels
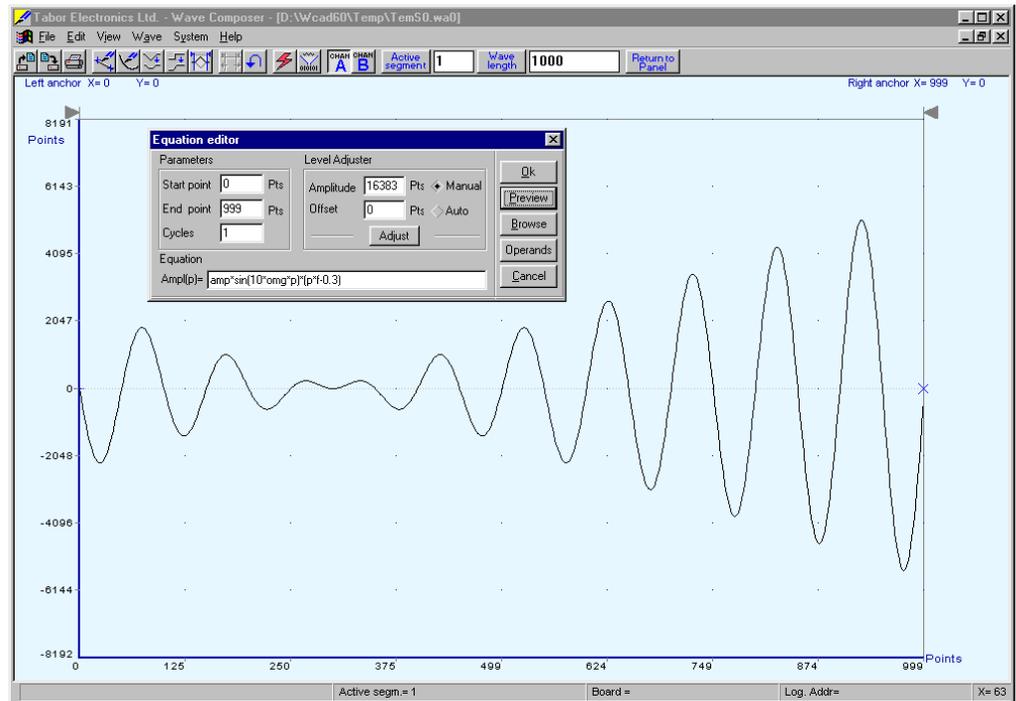
Figure 1-3, ArbExplorer - Wave Composer
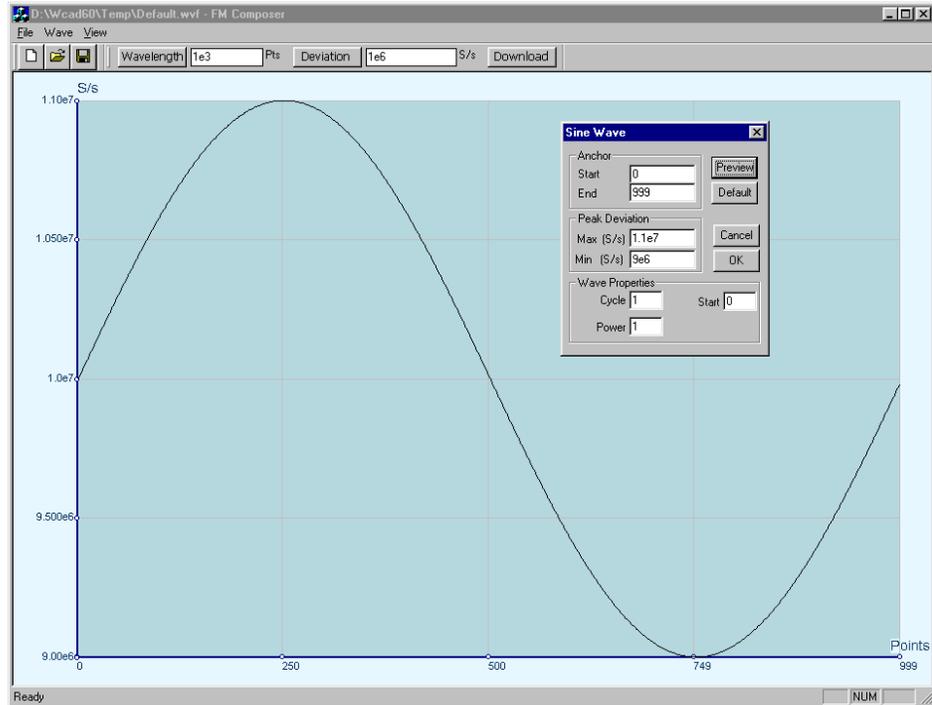
Figure 1-4, ArbExplorer - FM Wave Composer

Description

Detailed functional description is given following the general description of the features, functions, and options available with the 8026.

Two Channels The 8026 is a dual channel arbitrary waveform generator. Although the two channels share a single sample clock source, each channel can be controlled separately to generate different waveform, amplitude and sequence. Having a single sample clock source is an advantage in a dual-channel configuration because it allows tight control over inter-channel synchronization and leading edge start phase where the initial skew between the two channels is just a few nanoseconds.

Output Functions Model 8026 is completely digital. There are no analog functions resident in its hardware circuits. Data has to be downloaded to the instrument for it to start generating waveforms. The instrument can generate a few standard functions such as sine wave, triangular wave and square wave. Each time that a standard function is required, the instrument calculates its coordinates and places them in the waveform memory. Therefore, every time a standard function is selected, minimal time is required for the controller to compute the function and load its data to the waveform memory.

Frequency Waveform frequency is programmed with 10 digits. Frequency accuracy of the output waveform is determined by the clock reference. The internal reference oscillator provides 100ppm accuracy and

stability over time and temperature. If higher accuracy and/or stability are required, you may order the 8026 with 1ppm reference, or connect an external frequency reference to the rear-panel reference input connector.

Amplitude The output level may be programmed separately for each channel within from 20 mV to 20 Vp-p into an open circuit, or 10 mV to 10 V into 50Ω. Offset may be applied to the output to shift the signal either positive or negative. Offset and amplitude are inter-related, so make sure you understand the offset-amplitude ranges before you apply offset to your signal. A special fine offset generator is built into each channel, which allows programming of extremely small offset increments. This feature extends the offset resolution to 6 digits and is very useful in applications such as mixer balancing where few micro-volts could sway the balance either way.

Trigger Modes Besides its normal continuous mode, the 8026 responds to a variety of trigger sources. The output waveform may be gated, triggered, or generate a counted burst of waveforms. A built-in trigger generator, having a programmable period can be used as a replacement of an external trigger source. The internal trigger generator can be programmed with resolution of 7 digits.

Arbitrary Waveforms The Model 8026 generates arbitrary waveforms with 14 bits of vertical resolution. Any waveform it generates must first be downloaded to waveform memory. The arbitrary waveform memory is a bank of 14-bit words. Each word represents a point on the horizontal waveform scale. Each word has a horizontal address that can range from 0 to 1,048,576 (4,191,280 with the 4 Meg option installed) and a vertical address that can range from -8192 to +8191 (14 bits). Using a high speed clocking circuit, the digital contents of the arbitrary waveform memory are extracted and routed to the Digital to Analog Converter (DAC). The DAC converts the digital data to an analog signal, and the output amplifier completes the task by amplifying or attenuating the signal at the output connector.

Memory Segmentation There is no need to use the complete memory every time an arbitrary waveform is generated. Waveform memory can be divided into up to 4096 smaller segments and different waveforms can be loaded into each segment. The various segments may then be loaded into a sequence table to generate long and complex waveforms. The sequence table can link up to 4096 segments, while each segment can loop up to 1 million times.

Remote Control The instrument must be used in conjunction with a host computer. All of its functions, modes and parameters are fully programmable using SCPI commands and syntax. There are three ways to program the Model 8026, the first being low-level programming of each individual parameter using SCPI commands. The second alternative is to use ArbExplorer for high-level programming. ArbExplorer is a software package supplied with the 8026 that simulates a mechanical front

panel. It has all the necessary push buttons, displays and dials to operate the instrument as if you were using it on the bench. The third alternative is using application specific drivers, such LabVIEW.

The 8026 must be programmed to generate waveforms. Therefore, it is recommended that the user becomes familiar with its basic features, functions and programming concepts as described in this and the following chapters.

Frequency Agility The instrument generates its sample clock from a DDS circuit (direct digital synthesis). The DDS circuit enables frequency agility through the complete frequency range of the 8026. Having such an enormous range opens the door for a wide range of applications such as wide band sweep, FSK and frequency modulation. The 8026 can generate FSK, Ramped FSK and Linear or Logarithmic sweep. The instrument can also frequency modulate its carrier using one of its built-in waveforms, or with any user-defined modulating signal, which can be downloaded using the FM wave composer.

Multi-Instrument Synchronization There are applications requiring more than 2 synchronized channels, at time even 10 channels. Synchronization between completely independent, free-running instruments is not an easy task; Besides distribution of the sample clock to all instruments, each instrument has to be told when to start generating waveforms so that all start at the same point and with the same phase. Triggering multiple instruments to achieve synchronization is not enough because it will generate a jitter of ±1 count. Multi-instrument synchronization is built into the 8026, where special connectors and cables were designed to eliminate jitter and to provide start phase control. There is not limit to the number of 8026's that can be daisy-chained, multiple instrument synchronization is built into each instrument.

Options

There is one option available with the 8026. This option can not be retrofitted later. The 8026 must be ordered with the option installed.

Option 4 Meg – The basic 8026 comes with 1 Meg memory. Option 4 Meg expands memory size to 4 Meg.

Safety Considerations

The instrument has been manufactured according to international safety standards. The instrument meets EN61010, VDE 0411/03.81 and UL 1244 standards for safety of commercial electronic measuring and test equipment for instruments with an exposed metal chassis that is directly connected to earth via the chassis power supply cable.

WARNING

Do not remove instrument covers when operating or when the power cord is connected to the mains.

Any adjustment, maintenance and repair of an opened, powered-on instrument should be avoided as much as possible, but when necessary, should be carried out only by a skilled person who is aware of the hazard involved.

Supplied Accessories

The instrument is supplied with an Instruction Manual. The manual includes a disk with ArbExplorer for Windows 95/98/NT. Service Manual and inter-instrument synchronization cables are available upon request.

Specifications

Instrument specifications are listed in Appendix A. These specifications are the performance standards or limits against which the instrument is tested. Specifications apply under the following conditions: output terminated into 50Ω after 30 minutes of warm up time, and within a temperature range of 20°C to 30°C. Specifications outside this range are degraded by 0.1% per ^{o}C.

Functional Description

A detailed functional description is given in the following paragraphs. The description is divided into logical groups: front and rear panel connectors, operating modes, output type, output state, filters, synchronization, and front panel indicators.

Front Panel Connectors

The 8026 has 3 BNC connectors on its front panel: one main output for each channel and one SYNC output. These connectors are described below.

Channel 1 Output

The channel 1 output connector outputs fixed (standard) waveforms to 50 MHz, user (arbitrary) and sequenced waveforms with sampling clock to 100 MS/s. Output impedance is 50Ω, that is, the cable connected to this output should be terminated with 50Ω. Output amplitude accuracy is calibrated when connected to a 50Ω load. The output amplitude is doubled when the output impedance is above 1 MΩ.

Channel 2 Output

The channel 2 output connector outputs fixed (standard) waveforms to 50 MHz, user (arbitrary) and sequenced waveforms with sampling clock to 100 MS/s. Output impedance is 50Ω, that is, the cable connected to this output should be terminated with 50Ω. Output amplitude accuracy is calibrated when connected to a 50Ω load. The output amplitude is doubled when the output impedance is above 1 MΩ.

SYNC Output

The SYNC output generates a single TTL pulse for synchronizing other instruments (i.e., 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 along the waveform is programmable. The SYNC output is also used as marker output when the sweep function is turned on.

Front Panel Indicators

There are three LEDs on the front panel. When the output state is on, the light illuminates. There are two LEDs, one for each channel and one LED for the SYNC output.

Rear Panel Connectors

The 8026 has a number of connectors on its rear panel. These connectors are described below. Figure 1-5 shows rear panel plugs, connectors and other parts.
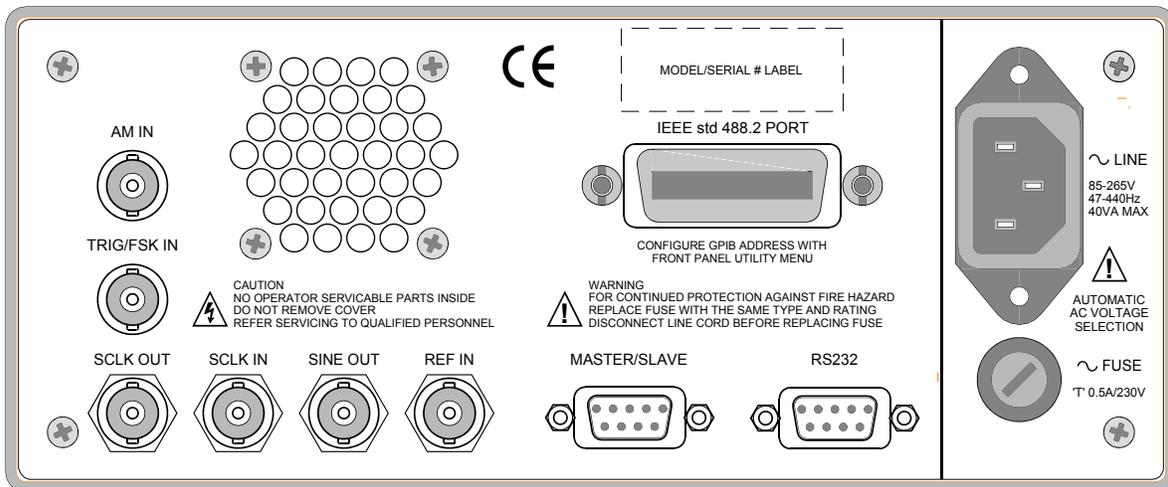
Figure 1-5, Model 8026 – Rear Panel

AM IN

This input accepts signals that amplitude modulate channel 1, 2, or both signal. The AM input becomes active only when the AM function is selected otherwise, it has no effect on the output signal.

TRIG/FSK IN

In general, this input accepts signals that stimulate generation of output waveforms. The trigger input is inactive when the generator is in continuous operating mode. When placed in trigger, gated or burst mode, the trigger input is made active and waits for the right condition to trigger the instrument. In trigger and burst modes, the trigger input is edge sensitive, i.e., it senses transitions from high to low or from low to high to trigger the 8026. The direction of the transition is programmable. In gated mode, the trigger input is level sensitive, i.e., the generator is gated when the logic level is high and idle when the level is logic low. Trigger level for this input is TTL.

The trigger input is common to both channels. Therefore, if the 8026 is placed in trigger mode, both channels share the same mode and the trigger input causes both channels to start generating waveforms at the same time. Phase relationship between channels is tightly controlled in trigger mode. You should expect both channels to start generating waves with exactly the same start phase. Further control of leading edge offset between channels is also provided.

The same input is used in FSK mode, where the output hops between two frequencies – carrier and shifted frequencies. The output generates carrier frequency when the FSK input is false and shifted frequency when the FSK input is true.

The trigger input is also used as stop and start input when the 8026 is placed in Arm mode.

SCLK OUT

This BNC connector outputs the programmed sample clock frequency. Output level is ECL, terminated into 50Ω to $-2V$. Note that correct termination is necessary for this output otherwise you will not see this signal at all. This output generates sample clock waveforms continuously, regardless if the 8026 is operating in continuous, trigger, or gated modes.

The sample clock output is used for multiple-instruments synchronization. In master mode, connect this output with a BNC to BNC connector to the adjacent slave instrument. You may also use this output to synchronize other components in your system to one master clock.

SCLK IN

This BNC connector accepts ECL level signal, terminated into 50Ω to $-2V$. The external clock input is available for those applications

requiring synchronization to one sample clock source. Normally, this input is disabled. When enabled, the clock at this input replaces the internal clock generator and the 8026 generates waveforms having the external sample clock rate.

The sample clock output is also used for multiple-instruments synchronization. In slave mode, connect this input with a BNC to BNC connector to the adjacent master instrument.

SINE OUT

This BNC connector outputs dc coupled, fixed level (1 V_{pk-pk} into 50Ω) sine waveforms. This output is derived directly from the sample clock generator and is active at all times, regardless of present operating mode of the 8026. Therefore, the frequency of the sine output is programmed using the sample clock parameter. Frequency agility and modulation affect this output directly. The sine waveform output is programmed to 100 MHz, so it may serve as an additional output to those available on the front panel.

REF IN

This BNC connector accepts 10 MHz, TTL level reference signal. The external reference input is available for those applications requiring better accuracy and stability reference than the one provided inside the 8026. The reference input is active only after selecting the external reference source mode.

MASTER/SLAVE

This 9-pin "D" connector is used when connecting master to slave units. The master/slave cable may be ordered separately from your nearest Tabor dealer. The master/slave mode must be activated from the front panel otherwise, this connector and the cable attached has no affect on normal operation of the 8026.

RS232

This 9-pin connector accepts standard RS232 cable. The 8026 operates at a fixed baud rate of 115K only. Connection and RS232 operation is explained later in this manual.

IEEE-488.2 PORT

This 24-pin connector accepts standard GPIB cable. The GPIB address is con figured using the front panel utility menu. The 8026 conforms to the IEEE-488.2 standard. Programming protocol is SCPI version 1993.0. GPIB cables are available separately from your Tabor dealer.

AC LINE

This 3-prong AC LINE connector accepts ac line voltage. The 8026 senses the line voltage and sets the appropriate range automatically. Therefore, traditional line voltage selector is not available on the rear panel. To avoid potentially hazardous situations, always connect the center pin to mains ground using the line cord that is supplied with the

instrument.

AC FUSE

The AC fuse protects the 8026 from excessive current. Always replace the fuse with the exact type and rating as printed on the rear panel. If the fuse blows again after replacement, we recommend that you refer your instrument immediately to the nearest Tabor service center.

Operating Modes

The 8026 can be programmed to operate in one of four operating modes: continuous, triggered, gated and counted burst. These modes are described below. As was discussed above, both channels are placed simultaneously in triggered mode. All channels will start generating waveforms when a valid trigger signal is received at the trigger input.

If you are using the one of the standard function, both waveforms will start and complete at exactly the same instance. In arbitrary mode, you may select different wavelength for each channel so, if you have different waveform length, the trigger will initiate one output cycle and the outputs will generate one complete waveform. The 8026 will idle when the longest waveform has completed.

Continuous Mode

In normal continuous mode, the selected waveform is generated continuously at the selected frequency, amplitude and offset.

Triggered Mode

In triggered mode, the 8026 circuits are armed to generate one output waveform. The trigger circuit is sensitive to transitions at the trigger input. Select between positive or negative transitions to trigger the instrument. When triggered, the generator outputs one waveform cycle and remains idle at the last point of the waveform. The instrument can be armed to receive a trigger signal from either the front panel connector, GPIB command, or from an internal, trigger generator. The internal trigger generator is programmable with 7 digits.

The trigger signal, whether it comes from the front panel or from a GPIB command, has to pass through some electrical circuits. These circuits cause small delay known as system delay. System delay cannot be eliminated completely and must be considered when applying a trigger signal. It defines how long it will take from a valid trigger edge to the moment that the output reacts.

Gated Mode

In gated mode, the 8026 circuits are armed to generate output waveforms as long as a gating signal is present. Unlike the triggered mode, the gated mode is level sensitive. When the gating signal goes low, the waveform at the output connector is first completed and the output reverts to an idle state. The idle amplitude level, after the

gating signal goes low, is the last point on the waveform.

Burst Mode

The burst mode is an extension of the triggered mode where the generator can be programmed to output a pre-determined number of waveforms. The source to trigger the burst can be selected from a front panel connector, GPIB trigger or from the built-in trigger generator.

Frequency Agility

Using the latest DDS (direct digital synthesis) technology, the 8026 is extremely agile. Operations like sweep, FSK and FM are directly derived from the DDS circuit by controlling its input bits. Frequency agility is described below.

Sweep

The 8026 can sweep from minimum to maximum sample clock frequency setting. You may select to sweep up or down using linear or logarithmic steps. Sweep time is programmable from 1 ms to 1000 seconds with 7 digits. The sweep start and stop frequency program the rear-panel sine output connector. You may also use the sweep from the front panel as long as you do your own calculation of start and stop frequencies, depending on the present sample clock frequency and waveform length. You may use the sweep in continuous mode, triggered, or gated. The 8026 will also sweep up, or down, using linear or logarithmic increments.

FSK

FSK (frequency shift keying) function controls the sine output connector. The trigger input is used to flag the 8026 when to output carrier frequency (trigger false) or when it should switch to the shift frequency (trigger true). You may also use the FSK function from the front panel as long as you do your own calculation of carrier and shifted frequencies, depending on the present sample clock frequency and waveform length.

Ramped FSK

Ramped FSK is the same as the FSK function except the output is ramped instead of switched to the shift frequency. Ramp time is programmable with 3 digits from 100 μs to 1 seconds.

FM

The FM function modulates the 8026 sample clock frequency. You can frequency modulate the output either with built-in waveforms, or download complex waveforms to the modulation memory. Using the latest DDS technology, the modulation is wide band and extremely linear. FM can be used in continuous, triggered and gated modes. The 8026 can generate two types of frequency modulation: 1) Standard and 2) Arbitrary. In standard mode, the modulating waveform is selected from a built-in library of 4 standard waveforms: sine, triangle, square and ramped wave shapes. In arbitrary mode, the modulating signal is downloaded to the modulation waveform memory. There are 20,000 points allocated specifically for the

arbitrary memory that is used for arbitrary frequency modulation.

Output Type

The 8026 can output three types of waveforms: standard (Fixed), arbitrary (User) and sequenced waveforms. Different waveform types may be assigned to each channel. Description of the various waveform types that the instrument can generate is given below.

Standard Waveforms

The 8026 must pre-load its memory before it can generate waveforms. On power up, the waveform memory has no specific data. The sine waveform, being the default waveform on power on, is computed and loaded to the waveform memory as part of the reset procedure. From this moment on, every time that another standard waveform is selected, it is being computed and loaded to the waveform memory.

Waveforms are written from the same start address. Therefore, every time that a new waveform is selected, there is some minimal time for the processor to compute and download the data to the memory.

The 8026 can be programmed to output one of nine standard waveform shapes: sine, triangle, square, pulse/ramp, sine(x)/x pulse, gaussian pulse, rising/decaying exponential pulse, noise and dc. There are some parameters associated with each waveform, which modify the shape of the waveform to better suit your needs. For example, different start phase for the sine waveform can be programmed for each channel to create phase offsets between the two output channels.

Arbitrary (User) Waveforms

The arbitrary waveform memory is capable of storing one or more user-defined waveforms. As was discussed before, and unless ordered differently, the 8026 is supplied with 1 Meg memory banks for each channel. With the 4 Meg option, there are up to 4 Meg points that can be allocated to one single waveform. On the other hand, there is no need to use the entire memory for only one waveform. The memory can be divided into smaller segments and loaded with different waveforms. The instrument can be programmed to output one segment at a time.

Loading data to arbitrary waveform memory can be a time-consuming task, especially if all 4 Meg points are loaded in one shot. The 8026 utilizes a DMA (direct memory access) concept that speeds data transfer from host computer to the instrument. In this mode, the memory bank is disconnected from the CPU circuit and re-routed to the GPIB bus for direct memory accessing by the host computer. This feature is available with the GPIB but not with the RS232.

There are two separate memory banks for each channel that can be loaded with different waveforms. Channels are not limited by the

number of segments or by the shape of the waveforms.

Sequenced Waveforms

The sequence generator is a powerful tool that lets you link and loop segments in any way you desire. As a simple example of a sequenced waveform, look at Figures 1-6a through 1-6c. The waveforms shown in these figures were placed in memory segments 1, 2 and 3, respectively. The sequence generator takes these three waveforms links and loops them in a predefined order to generate the waveform shown in Figure 1-6d.

The sequence circuit is useful for generating long waveforms with repeated sections. The repeated waveform has to be programmed once and the repeater loops on this segment as many times as selected. When in sequenced mode, there is no loss of time between linked or looped segments.
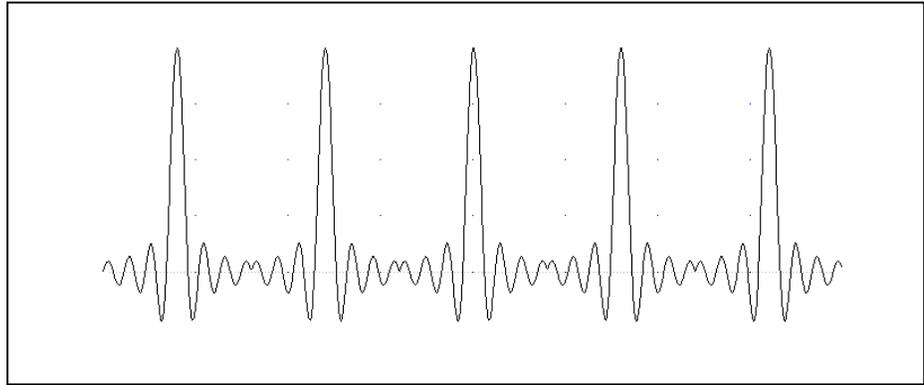
Figure 1-6a , Segment 1 – Sin (x)/x Waveform
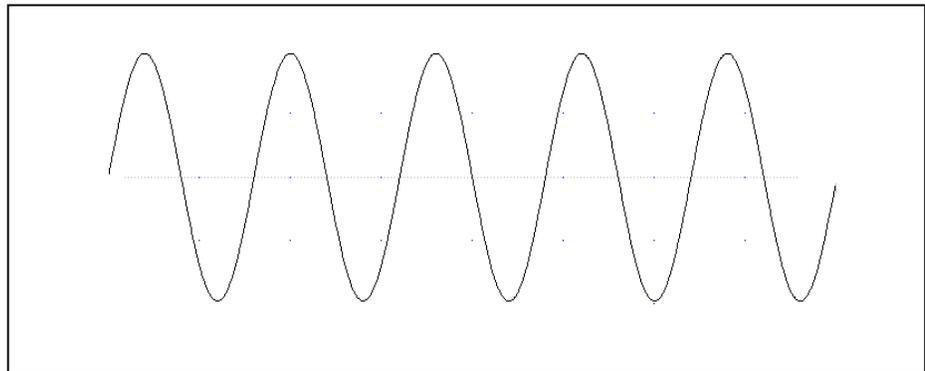
Figure 1-6b. Segment 2 – Sine Waveform
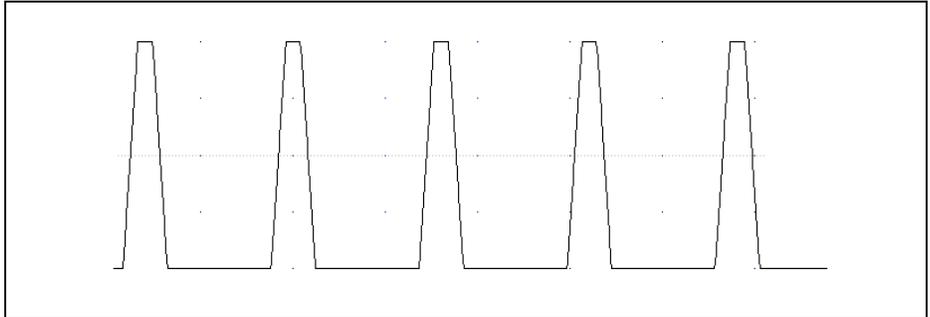
Figure 1-6c Segment 3 – Pulse Waveform

The following sequence was made of segment 2 repeated twice, segment 1 repeated four times, and segment 3 repeated two times.
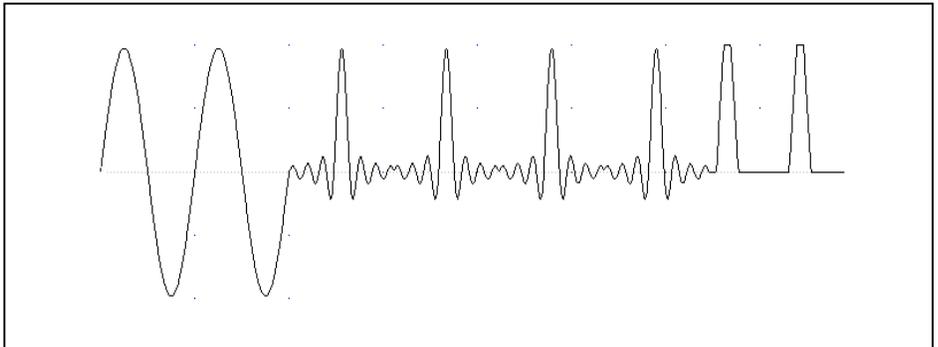
Figure 1- 6d. Sequenced Waveforms

Output State

The main outputs can be turned on or off. The internal circuit is disconnected from the output connector by a mechanical switch (relay). This feature is useful for connecting the main outputs to an analog bus. For safety reasons, when power is first applied to the chassis, the main output is always off.

Filters

Two filters for each channel are built into the 8026. These filters are available for use in various applications such as the creation of high frequency sine waves and removing the staircase effect from waveforms that are generated with high frequency clock rates. The filters are also used for reconstructing the standard sine waveform.

Programming The 8026

The 8026 has front panel control capability for most of its function, parameters, and modes. However, waveform data and sequence tables must be loaded to the 8026 from a host computer before it can be output arbitrary or sequenced waveforms. There are a number of ways to "talk" to the instrument. They all require that an appropriate software driver be installed in the host computer. The rest is a matter of practice and knowledge of the language in use. There are other system considerations like address selection that have to be settled before programming the instrument. These topics are discussed in later chapters.

Low level programming of the 8026 is accomplished using SCPI (Standard Commands for Programmable Instruments) language. Programming aspects are covered in Chapters 3 and 4.

Supplied with the 8026 is a PC software package called ArbExplorer. This provides a user interface with a familiar windows interface which allows the user to interact with and control the 8026 directly. Details of how to use ArbExplorer are supplied in the following pages.

Chapter 2

CONFIGURING The INSTRUMENT

Installation Overview

This chapter contains information and instructions necessary to prepare the Model 8026 for operation. Details are provided for initial inspection, grounding requirements, repackaging instructions for storage or shipment and installation information.

Unpacking and Initial Inspection

Unpacking and handling of the generator requires normal precautions and procedures applicable to handling of sensitive electronic equipment. The contents of all shipping containers should be checked for included accessories and certified against the packing slip to determine that the shipment is complete.

Safety Precautions

The following safety precautions should be observed before using this product and associated computer. Although some instruments and accessories would normally be used with non-hazardous voltages, there are situations where hazardous conditions may be present.

This product is intended for use by qualified persons who recognize shock hazards and are familiar with the safety precautions required to avoid possible injury. Read the operating information carefully before using the product.

WARNING

For maximum safety, do not touch the product, test cables, or any other instrument parts while power is applied to the circuit under test. ALWAYS remove power from the entire test system before connecting cables or jumpers, installing or removing cards from the computer, or making internal changes such as changing the GPIB address. Do not touch any object that could provide a current path to the common side of the circuit under test or power line (earth) ground. Always keep your hands dry while handling the instrument.

Exercise extreme caution when a shock hazard is present. Lethal voltage may be present on cables, connector jacks, or test fixtures. The American National Standard Institute (ANSI) states that a shock hazard exists when voltage levels greater than 30V RMS, 42.4V peak, or 60VDC are present. Before operating an instrument, make sure the line cord is connected to a properly grounded power receptacle. Inspect the connecting cables and test leads for possible wear, cracks, or breaks before each use.

For maximum safety, do not touch the product, test cables or any other of the instrument parts while power is applied to the circuit under test. ALWAYS remove power from the entire test system before connecting cables or jumpers, installing or removing cards from the computer or making internal changes, such as changing card address.

Do not touch any object that could provide a current path to the common side of the circuit under test or power line (earth) ground. Always keep dry hands while handling the instrument. When using test fixtures, keep the lid closed while power is applied to the device under test. Carefully read the "Safety Precautions" instructions that are supplied with your test fixtures.

Before performing any maintenance, disconnect the line cord and all test cables. Only qualified service personnel should perform maintenance. If you have no past experience in computer servicing, we strongly recommend that installation and initial tests on the instrument be done by your computer dealer or by the factory itself

Performance Checks

The instrument has been inspected for mechanical and electrical performance before shipment from the factory. It is free of physical defects and in perfect electrical order.

Power Requirements

The function generator may be operated from a wide range of mains voltage – 85 to 265 Volts. Voltage selection is automatic and does not require switch setting. The instrument operates over the power mains frequency range of 48 to 440Hz. Always verify that the operating power mains voltage is the same as that specified on the rear panel.

The 8026 should be operated from a power source with its neutral at or near ground (earth potential). The instrument is not intended for operation from two phases of a multi-phase ac system or across the legs of a single-phase, three-wire ac power system. Crest factor (ratio of peak voltage to rms.) should be typically within the range of 1.3 to 1.6 at 10% of the nominal rms. mains voltage.

Grounding Requirements

To ensure the safety of operating personnel, the U.S. O.S.H.A. (Occupational Safety and Health) requirement and good engineering practice mandate that the instrument panel and enclosure be "earth" grounded. The long offset pin on the male end of the power cable carries the ground wire to the long pin of the Euro connector (DIN standard) receptacle on the rear panel of the instrument.

To preserve the safety protection feature when operating the instrument from a two-contact outlet, use a three-prong to two-prong adapter and connect the green lead on the adapter to an "earth" ground.

WARNING

Do not attempt to float the output from ground as it may damage the Model 8026 and other equipment connected to the 8026 connectors.

CAUTION

To avoid operator shock hazard do not exceed the power mains voltage frequency rating which limits the leakage current between case and power mains. Never expose the instrument to rain, excessive moisture, or condensation.

Long Term Storage or Repackaging For Shipment

If the instrument is to be stored for a long period of time or shipped immediately, proceed as directed below. If you have any questions, contact your local Tabor representative or the Tabor Customer Service Department.

1. Repack the instrument using the wrappings, packing material and accessories originally shipped with the unit. If the original container is not available, purchase replacement materials.

2. Be sure the carton is well sealed with strong tape or metal straps.

3. Mark the carton with the model and serial number. If it is to be shipped, show sending and return address on two sides of the box.

NOTE

If the instrument is to be shipped to Tabor for calibration or repair, attach a tag to the instrument identifying the owner. Note the problem, symptoms, and service or repair desired. Record the model and serial number of the instrument. Show the work authorization order as well as the date and method of shipment. ALWAYS OBTAIN A RETURN AUTHORIZATION NUMBER FROM THE FACTORY BEFORE SHIPPING THE INSTRUMENT TO Tabor.

Preparation For Use

Preparation for use includes removing the instrument from the container box, selecting the required GPIB address, connecting BNC cables to the 8026 input and output connectors and connecting the instrument to the power line and to a host computer.

Logical Address Selection

The GPIB Resource Manager identifies instruments in the system by the instrument's address. GPIB addresses can range from 0 to 31, however, addresses 1 to 31 **only** are reserved for GPIB instruments. Address 0 is reserved for the GPIB controller.

To change the GPIB address, press the UTILITY button and then press the button pointing to the GPIB address field. Note the cursor under the GPIB address. Rotate the dial until the 8026 displays the required address. Tabor ships the Model 8026 with GPIB address set to 4.

Installation

The instrument is fully solid state and dissipates only a small amount of power. No special cooling is required. However, the instrument should not be operated where the ambient temperature exceeds 40°C, when the relative humidity exceeds 80% or condensation appears anywhere on the instrument.

Avoid operating the instrument close to strong magnetic fields, which may be found near high power equipment such as motors, pumps, solenoids or high power cables. Use care when rack mounting to locate the instrument away from sources of excessive heat or magnetic fields. Always leave 4 cm (1.5 inches) of ventilation space on all sides of the instrument.

Bench Operation

The Model 8026 is shipped with plastic feet, front tilt stands in place and ready for use as a bench or portable instrument

Rack Mounting

The instrument may be rack mounted in a standard 19-inch rack. Rack mounting ears are available as an option.

USING The INSTRUMENT

Overview

This chapter contains information about how to operate the 8026. Operation is divided into two general categories: basic bench operation, and remote operation (IEEE-488.2 and RS232). Basic bench operation, which is covered in this section, describes how to use the arbitrary waveform generator using front panel sequences. The 8026 is supplied with ArbExplorer, a PC based software package with a graphical user interface to allow users to program all of the functions directly. LabView drivers and a set of SCPI commands are available for more experienced programmers.

The following paragraphs describe the various modes of operation and give examples of how to program the Model 8026. The manual is organized by instrument function and instructions are given in each paragraph on how to use the function from both the front panel and ArbExplorer.

Inter-Channel Dependency

The 8026 has two output channels. Although this is a two-channel instrument, many of the commands that set parameters and functions are common for both channels. For example, sample clock and trigger modes can not be set separately for each channel. On the other hand, you may set each channel to have different function shape, amplitude and offset. Table 3-1 lists the function and parameters and their related Inter-channel dependency.

Output Termination

During use, output connectors must be properly terminated to minimize signal reflection or power loss due to impedance mismatch. Proper termination is also required for accurate amplitude levels at the output connectors. Use 50Ω cables and terminate the main and SYNC cables with terminating resistors. Always place the 50Ω termination at the far end of the cables.

Input / Output Protection

The Model 8026 provides protection for internal circuitry connected to input and output connectors. Refer to the specifications in Appendix A to determine the level of protection associated with each input or output connector.

Power On/Reset Defaults

The 8026 utilizes non-volatile memory backup that automatically stores the last setup before the generator was turned off. Every time you turn on the instrument, the non-volatile memory updates the front panel setting with modes, parameters and waveforms with its last setting with only one exception, for safety reasons, the outputs remain off even if they were turned on before powering down the 8026.

After power on, the instrument displays information messages, as shown below in Figure 3-1. Press any button on the front panel to access modes or change parameters.
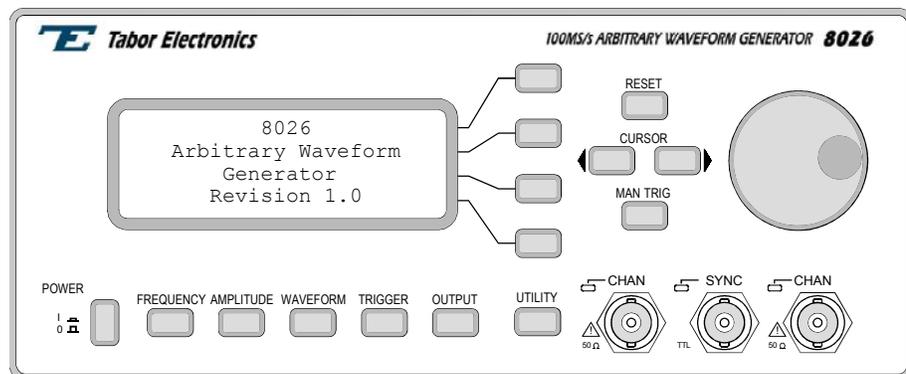
Figure 3- 1. Power on Display

If you are not yet fully familiar with front panel operation of the 8026, you may find yourself locked into a "dead-end" situation where nothing operates the way it should. The fastest way to restore the generator to a known state is by resetting the instrument to factory defaults.

Observe Figure 3-2 and reset parameters to factory defaults as follows:

1. Press the RESET button. The instrument displays two options

2. Press button 2 to reset just one parameter, or

3. Press button 3 to restore factory defaults

Table 3-1 summarizes factory defaults for the most common parameters. A complete list of all parameters, their defaults, as well as their maximum and minimum values is given in Chapter 4.
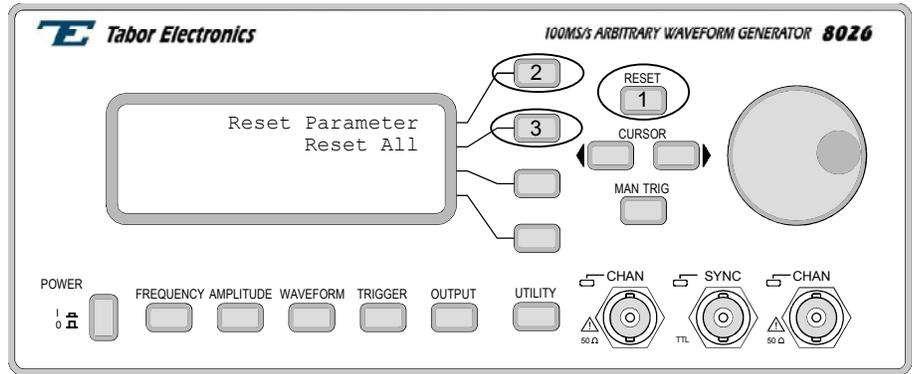
Figure 3- 2. Reset Options.

Function / Parameter	Default	Inter-Channel Dependency
Output State:	Off	Separate
Operating Mode:	Continuous	Common
SYNC State:	Off	Common
Output Function:	Standard	Separate
Output Function Shape:	Sine	Separate
Standard Wave Frequency:	1 MHz	Common
User Wave Sample Clock:	10 MS/s	Common
Sample Clock Source & Reference:	Internal	Common
Channel 2 Clock Divider:	1	Separate
Amplitude:	5 V	Separate
Offset:	0 V	Separate
Filter State:	Off	Separate
Filter Type:	Auto	Separate
Trigger Slope:	Positive	Common
Internal Trigger Rate:	1 KHz	Common
Modulation State:	Off	Common
DMA State:	Off	Separate

Table 3-1. Default Conditions After Reset

Controlling the 8026

Controlling 8026 function, modes and parameters is simply a matter of pressing once or twice the appropriate button as described in the following paragraphs. Refer to Figure 3-3 throughout this description.
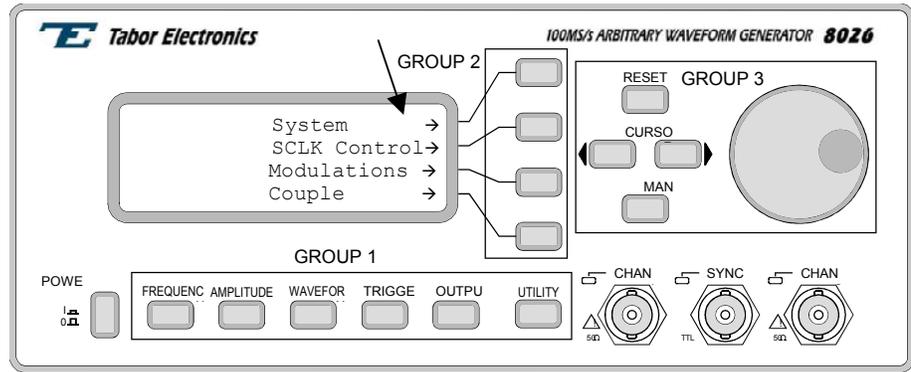
Figure 3-3. Front Panel Operation

In general, buttons are grouped to perform similar functions. Group 1 is used to access main menus, group 2 selects parameters or sub-menus and group 3 controls cursor position and parameter values. Use the same sequence of operation any time you want to select functions, change modes, or change parameters. A list of menus, sub-menus and parameters is summarized in Table 3-2.

NOTE

Notice the right-hand arrows next to an option. The arrow is there to show you that there is a submenu associated with this line. If you press the key in group 2, the display will automatically take you to the submenu.

Table 3-2. Front Panel Menus

Main Menu	Sub-Menu	Parameter	Notes
FREQUENCY			
	Freq	1.000000MHz	Waveform frequency
	Sclk	10.00000MS/s	Sample clock frequency
	Divider CH2	1	Channel 2 sclk divider

Table 3-3. Front Panel Menus (continued)

Main Menu	Sub-Menu	Parameter	Notes
AMPLITUDE			
	Ampl CH1	5.000V	Channel 1 amplitude
	Offs CH1	0.000V	Channel 1 offset
	Ampl CH2	5.000V	Channel 2 amplitude
	Offs CH2	0.000V	Channel 2 offset
WAVEFORM			
	STD.	CH1, CH2 waves	Standard waveforms
	ARB.	Segments	Arbitrary waveforms
	SEQ.	Advance modes	Sequence Advance Modes
TRIGGER			
	CONT	Arm parameters	Continuous
	TRIG	Trig parameters	Triggered
	GATED	Gate parameters	Gated
	BURST	Burst parameters	Counted burst
OUTPUT			
	Outputs	Output CH1	Channel 1 on/off
		Output CH2	Channel 2 on/off
		CH2->CH1 Offset	Inter-channel phase offset
	SYNC Out	Output	SYNC output on/off
		Type	Select bit, or lcom
		Position	SYNC position
		Width	SYNC pulse width
	Filters	Filter CH1	Select channel 1 filter
		Filter CH2	Select channel 2 filter
UTILITY			
	System	GPIB Address	Select GPIB address
		Selftest	Perform selftest
		Setups	Save/Recall front panel setups
	SCLK Control	Sclk Source	Select sample clock source
		Sclk Reference	Select sample clock reference
	Modulations	AM	Set AM on/off
		FM	Set FM parameters
		FSK	Set FSK parameters
		Sweep	Set sweep parameters
	Couple	Couple State	Set couple state on/off
		Couple Mode	Set couple mode master/slave
		Edge Offset	Set inter-instrument edge offset

Enabling the Outputs

The 8026 waveforms are generated using data points that are downloaded to a waveform memory. Using the sample clock generator, the data is clocked to the digital to analog converter (DAC) which, in turn, generates the waveform shape that was downloaded to the arbitrary waveform memory.

The DAC itself can not be connected directly to the main output because it does not have sufficient drive capabilities, nor does it have the amplitude and offset control, which is required at the output connector. To overcome this problem, a high-power output amplifier circuit is installed directly after the DAC. The amplifier circuit includes gain, offset, attenuation and stand-by control. The output amplifier is the circuit that connects to the output connector.

For safety reasons, main output default setting is OFF. Observe Figure 3-4 and disable or enable the main output using the following procedure.

1. Press the OUTPUT button
2. Select the Outputs menu
3. Select the channel you want to control
4. Toggle channel 1 or channel 2 outputs on and off using the dial
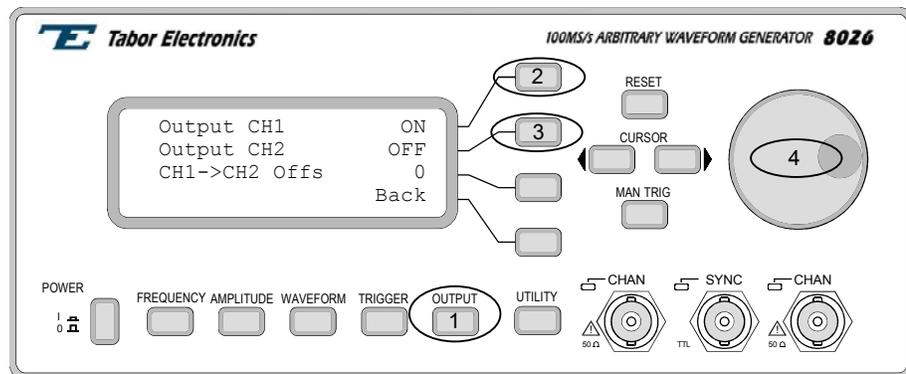
Figure 3-4. Enabling the Outputs

Selecting an Output Type

There are three main types of waveforms that the 8026 can produce: Standard (fixed), Arbitrary (user) and Sequenced waveforms. Standard waveforms are computed from equations and tables that are built into the program. The instrument can output arbitrary and sequenced waveforms only after waveform data has been downloaded into its memory. Refer to Figure 3-5 and use the following procedure to select an output type. Note that there are sub-menus associated with each output type menu. Accessing and using these sub-menus is described later in this chapter.

1. Press the Waveform button

2. Rotate the dial counterclockwise to select between STD (standard), ARB (arbitrary) and SEQ (sequenced) waveforms. The displayed waveform type is automatically available at the output connector.
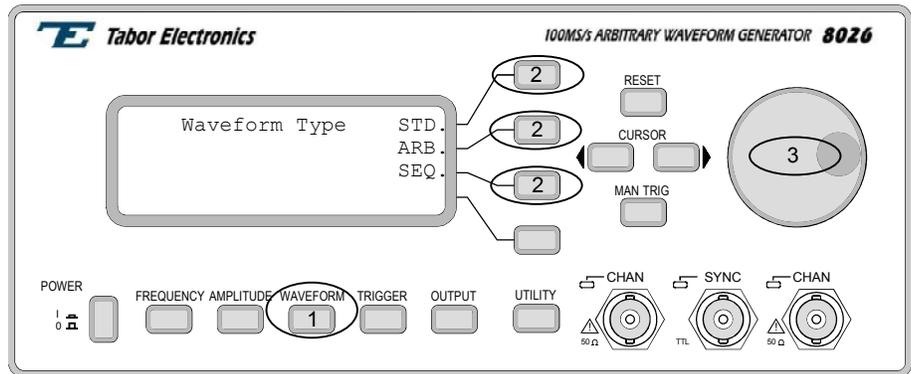
Figure 3-5. Selecting Waveform Type

Changing the Frequency, Sample Clock and Channel 2 SCLK Divider

The user should be careful not to confuse waveform frequency with sample clock frequency. The waveform frequency parameter is valid for standard waveforms only and controls waveform frequency at the output connector. On the other hand, the sample clock frequency parameter is valid for arbitrary waveforms only and defines the frequency of which the generator clocks data points.

Standard waveform frequency is measured in units of Hz. Arbitrary waveform sample clock frequency is measured in units of S/s (samples per second). The frequency of a given arbitrary waveform at the output connector is dependant on sample clock frequency, the number of data points, and the specifics of the waveform defined.

The sample clock divider is used to divide the sample clock for channel 2. This way, each channel can have a different sample clock rate. The sample clock is divided using integer numbers only.

Observe Figure 3-6 and modify frequency and sample clock using the following procedure.

1. Press the FREQUENCY button

2. Select Freq, Sclk, or Divider CH2 parameter

3. Move the cursor from left to right until it appears under the digit you want to change

4. Use the dial to modify the parameter. The output waveform is updated immediately as you rotate the dial however, note that

frequency has an affect on standard waveforms and sample clock and the divider parameters affect arbitrary and sequenced waveforms only.
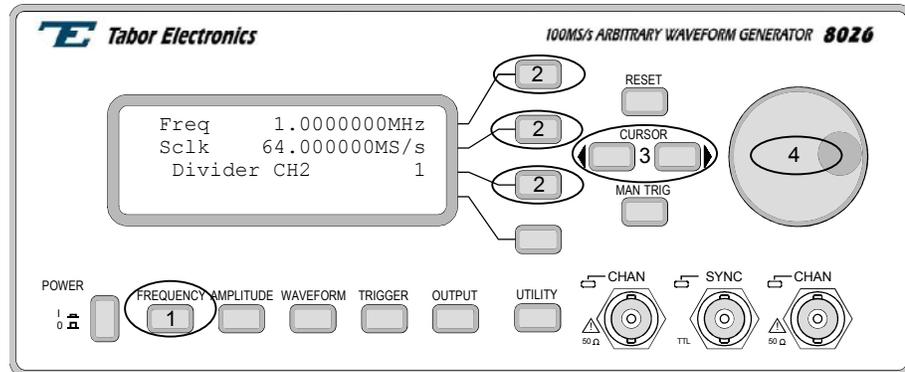
Figure 3-6. Modifying Frequency, Sample Clock and Channel 2 Sample Clock Divider

Programming the Amplitude and Offset

Output amplitude and offset can be programmed separately for each channel. Amplitude and offset are set within windows, so before you select values for these parameters, make sure that you do not exceed amplitude and offset limits.

The 8026 has fine offset adjustment, which expands the offset resolution to nearly 19 bits. The fine offset does not depend on amplitude-offset ranges so it could be applied at any value. For example, you have 4V amplitude and 1.100V offset. Fine offset setting of 50μV will give you an output offset of 1.10005V. This feature is extremely useful in applications where tiny null offsets are required, for example, balancing mixer inputs.

The relationship between amplitude and offset is defined using the following equation:

$$Window \geq \frac{Amplitude}{2} + |Offset|$$

Table 3-4 summarizes the various amplitude ranges and their respective amplitude windows and offset ranges.

Range Index	Amplitude Range	Basic 8026		Window
		Offset Range	Offset Resolution	
1	10.00V to 3.162V	±4.50000V	10μV	±5V
2	3.161V to 1.000V	±4.50000V	10μV	±5V
3	999.9mV to 316.2mV	±450.000mV	1μV	±500mV
4	316.1mV to 100.0mV	±450.000mV	1μV	±500mV
5	99.99mV to 31.62mV	±45.0000mV	0.1μV	±50mV
6	31.61mV to 10.00mV	±45.0000mV	0.1μV	±50mV

Table 3-4. Amplitude – Offset Inter-Relation

Notes

1. **While setting amplitude and offset, note that the Amplitude Range defines the Range Index. Legal offset values may be set using the appropriate Range Index. Front panel setting will automatically detect the appropriate range and will not allow error conditions. If you program amplitude and offset from remote and use two different Range Indexes, an error will occur.**

---

2. **Offset fine is an exception. You may exceed the window range with the offset fine and no error will result.**

Observe Figure 3-7 and modify amplitude and offset using the following procedure.

1. Press the AMPLITUDE button

2. Select Ampl CH1, Offs CH1, or Offs Fine CH1 parameter. Press More to access channel 2 amplitude and offset parameters.

3. Move the cursor from left to right until it appears under the digit you want to change

4. Use the dial to modify the parameter. The output waveform is updated immediately as you rotate the dial.
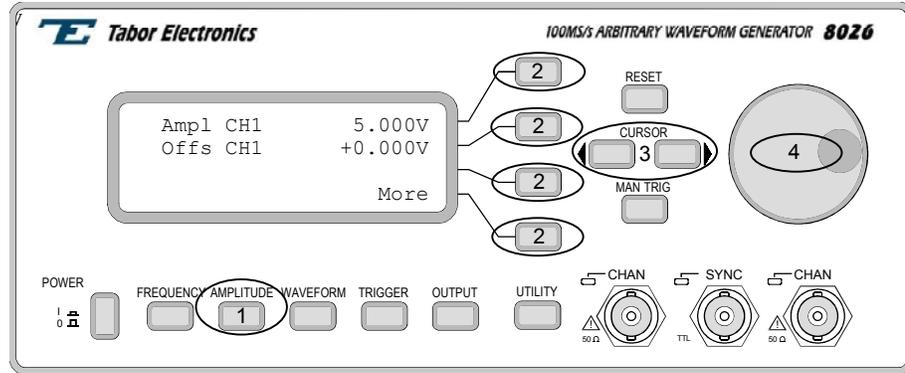
Figure 3-7. Modifying Amplitude, Offset and Offset Fine

Selecting a Standard Function Shape

Standard (fixed) waveforms are built into the 8026 program. Ten standard function shapes are available: Sine, Triangle, Square, Pulse, Ramp, Sinc, Exponential decaying pulse, Gaussian pulse, Noise and DC. Every time a standard function is selected the coordinates for this function are re-computed and placed in the waveform memory. Therefore, there is a minimal delay after you select the function until the output starts generating the waveform.

The number of points used to define each Standard Waveform is the same. Thus, some parameters may not have any effect on the waveform because too few points are available to generate the waveform. The number of points for each standard waveform at various frequencies is given below.

The number of points used for generating standard waveforms at various frequencies is computed as follows:

Frequency $\leq$ 97.6562 KHz, Points = 1024
Frequency > 97.6562 KHz, Points = N, where

$$N = 8 \cdot 2^{\; INT\left\{ \log_2 \left[\dfrac{INT\left(\dfrac{100}{FREQ} \right)}{8} \right] \right\}}$$

Each waveform shape has a number of variable parameters associated with it other than frequency and amplitude. Parameters, which may be changed and their units are given in Table 3-5. The mathematical expressions for the positive exponential pulse, negative exponential pulse, gaussian pulse, and sinc pulse are also given.

Waveform	Variable Parameter	Units
Sine	Start Phase	Degrees
Triangle	Start Phase	Degrees
Square	Duty Cycle	% of Period
Pulse	Delay	% of Period
	Rise Time	% of Period
	High Time	% of Period
	Fall Time	% of Period
Ramp	Delay	% of Period
	Rise Time	% of Period
	Fall Time	% of Period
Sinc	Zero Crossings	Integer
Gaussian	Time Constant	Integer
Exponential	Time Constant	Integer
Noise	None	
DC	Amplitude	% of Amplitude

Table 3-5. Standard Waveform Adjustable Parameters

Mathematical Expressions:

SINC
$$F(m) = \frac{Sine\left(2\pi\frac{m}{R}\right)A}{2\pi\frac{m}{R}}$$

R = N/cyc (number of points per cycle)
cyc = Number of SINC zero crossings

Exponential Rise
$$F(m) = -\frac{A}{2} + \frac{A}{2}\left[e^{\frac{m}{7.71}} - 1\right]$$

A = Amplitude
m = Current point (1..N)
N = Total number of points
t = Time constant set by user

Exponential Decay
$$F(m) = Ae^{-m/t} - \frac{A}{2}$$

Gaussian
$$F(m) = Ae^{-m^2/t^2} - \frac{A}{2}$$

Refer to Figures 3-8 and 3-9. Select standard waveform and modify parameters using the following procedure. Procedure is shown for

channel 1 waveforms. Use the same procedure to modify channel 2 waveforms. Figure 3-8 shows waveform shape selection and Figure 3-9 shows parameter adjustments. The example below describes how to select the pulse waveform.

1. Press the WAVEFORM button

2. Select and press the STD option

3. Rotate the dial until PULS is displayed

4. Press the PULS button to access the pulse parameters menu

5. Select the required parameter and press the pointing button

6. Move the cursor from left to right until it appears under the digit you want to change

7. Use the dial to modify the parameter. The output waveform is updated immediately as you rotate the dial.
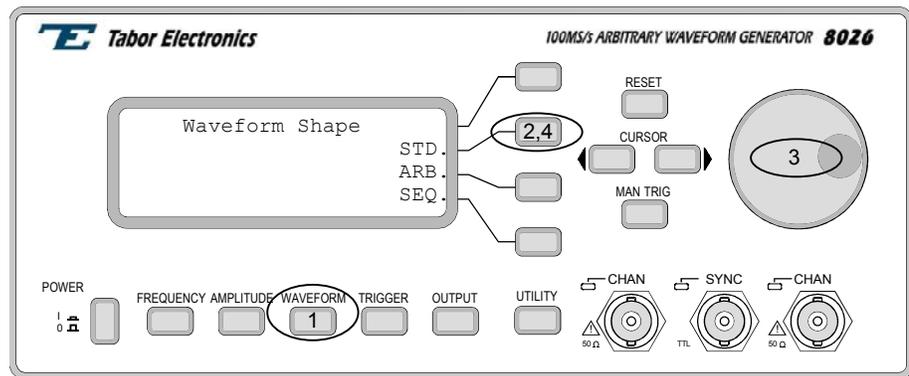
Figure 3-8. Selecting Standard Waveform Shape
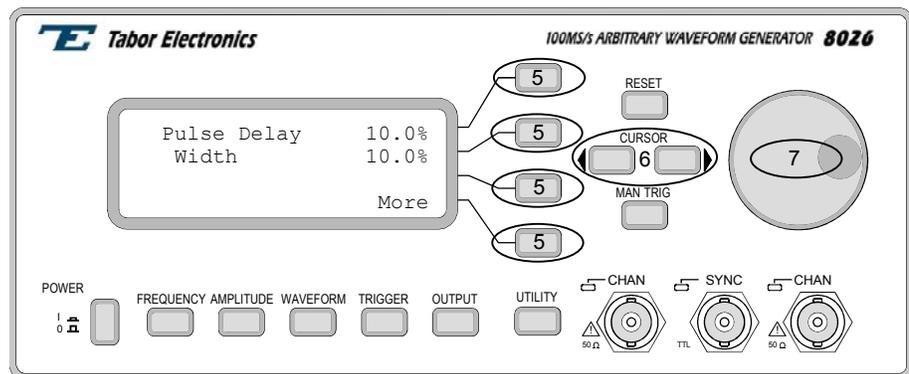
Figure 3-9. Adjusting Waveform Parameters

Selecting an Operating Mode

The Model 8026 offers four operating modes: Continuous, Triggered, Gated and Burst. The selected waveform is repeated continuously when the instrument is set to operate in Continuous mode. In this mode, the 8026 does not require a trigger source to stimulate output cycles. The operating mode defaults to continuos after reset.

Triggered, Gated, and Burst modes require an external signal to initiate output cycles. In some case, an internal trigger generator is available to generate the required trigger stimuli without the need to connect to external devices.

Information on how to operate the generator in trigger, gate or output a burst of waveforms is given in the following paragraphs.

Triggered Mode

In Triggered mode, the output remains at a zero DC level as long as the trigger signal to the rear-panel remains inactive. The trigger input is sensitive to either a rising edge or a falling edge. Each time a transition occurs at the trigger input, the 8026 generates one complete output waveform. At the end of the output cycle, the output resumes position at a DC level that is equal to the amplitude of the last point of the waveform. There are four parameters you can adjust for this mode:

1. Slope – Defines edge sensitivity for the trigger input
2. Source – Defines the trigger source. External enables the trigger input. Internal disables the trigger input and enables an internal free-running trigger generator that automatically triggers the output at programmable intervals
3. Timer – Set the intervals for the internal trigger generator, and
4. Start Phase – Defines the start phase for the trigger signal. The start phase parameter may serve as a trigger delay generator, where the delay is set in number of waveform points.

You may use the trigger mode to trigger standard, arbitrary and sequenced waveforms. Observe the limitations of the trigger signal as listed in the specification section of this manual.

Gated Mode

When set to gated mode, the 8026 output remains at a DC level as long as the rear-panel TRIG IN signal remains inactive. The gating signal can be programmed to be either active high or active low. The Model 8026 generates output waveforms as long as the proper level is present at the trigger input connector. When the gate signal is de-asserted, the output completes the last cycle and resumes position at a DC level equal to the last point of the waveform. There is only one parameter you can adjust for the gated mode:

1. Slope – Defines if the generator is gating when the trigger signal is TTL high (POS) or when the trigger signal is TTL low (NEG)

You may use the gated mode to gate standard waveforms, arbitrary waveforms, and sequences of waveforms. Observe the limitations of the gating signal as listed in the specification section of this manual.

Burst Mode Burst mode is similar to Triggered mode with the exception that only one trigger signal is needed to generate a counted number of output waveforms. In Burst mode, the output remains at a DC level as long as the trigger signal at the rear panel remains inactive. The trigger input is sensitive to either the rising edge or the falling edge. Each time a transition at the trigger input occurs, the 8026 generates predetermined number of waveform cycles. At the end of the counted burst, the output resumes position at a DC level equal to the last point of the waveform. The burst count is settable from 1 to 1 million cycles. There are four parameters you can adjust for this mode:

1. Slope – Defines edge sensitivity for the trigger input
2. Source – Defines the trigger source. External enables the trigger input. Internal disables the trigger input and enables an internal free-running trigger generator that automatically triggers the output at programmable intervals
3. Timer – Set the intervals for the internal trigger generator, and
4. Start Phase – Defines the start phase for the trigger signal. The start phase parameter may serve as a trigger delay generator, where the delay is set in number of waveform points.

You may use the counted burst mode in conjunction with standard and arbitrary waveforms only. Sequenced waveforms can not be used in burst mode.

Arm Mode The Arm mode allows you to stop generating waveforms at specific breakpoint location, then continuing with generating the waveform only when you are ready. The arm mode operates in continuous mode only. There are three parameters for this function:

1. State – Turns arm mode on and off
2. Slope – Defines edge sensitivity for the trigger input, and
3. Breakpoint Position – Defines where the waveform will stop

In continuous mode, the instrument generates waveforms continuously. Valid trigger transition at the trigger input will stop the waveform at the selected breakpoint position. The next trigger will enable continuous operation again and the waveform will start from the last breakpoint position. A remote trigger command can replace the external trigger input..

Figure 3-10 and Figure 3-11, along with the following description

demonstrate how to set the 8026 to internal trigger mode and then adjust the internal trigger timer to the required interval.

1. Press the TRIGGER button
2. Select and press the TRIG mode
3. Select and press Source.
4. Rotate the dial counterclockwise to select INT.
5. Press and select Timer.
6. Move the cursor from left to right until it appears under the digit you want to change
7. Use the dial to modify the parameter. The output waveform is updated immediately as you rotate the dial.
8. You may press More if you want to access the trigger start phase parameter.

Figure 3-10. Selecting a Trigger Mode

Figure 3-11. Programming trigger parameters

Using the Manual Trigger

The manual trigger allows you to trigger or gate the 8026 directly from the front panel. This button is active only when the generator is placed in external trigger, or arm modes. The MAN TRIG button has no affect on the instrument in other modes.

Using the SYNC Output

For safety reasons, every time you turn the 8026 OFF and ON, the SYNC output defaults to OFF. If you want to use the SYNC output you must turn it on immediately after you power up the generator.

When SYNC output generates two type of signals:

BIT   The BIT mode is recommended to be used in standard and arbitrary modes only. The sync output generates a single sync signal with every waveform period. As default, the sync signal is positioned at the beginning of the waveform. You also have the flexibility to control sync output parameters:

1. Position – Lets you place the sync bit at any point along the waveform. Placement resolution is 4 points.
2. Width – Programs the width of the sync signal as percent of the output period. Width resolution is 4 points.

LCOM   The LCOM (loop complete) option generates a sync signal in sequenced mode only. The signal starts with the selected segment and stops only after the sequence loop has completed. Position and width parameters are ignore when you select the LCOM type.

Use Figures 3-12, 3-13 and 3-14, along with the following description to learn how to set turn on the SYNC output, select the output type and adjust sync position and width.

1. Press the OUTPUT button
2. Select and press the SYNC Out
3. Select and press the Output
4. Rotate the dial clockwise to turn the output ON.
5. Select the More button
6. Press and select the Position.
7. Move the cursor from left to right until it appears under the digit you want to change
8. Rotate the dial to modify the sync position value. The sync output waveform is updated immediately as you rotate the dial
9. Press and select Width.
10. Move the cursor from left to right until it appears under the digit you want to change

11. Rotate the dial to modify the sync width. The sync output waveform is updated immediately as you rotate the dial
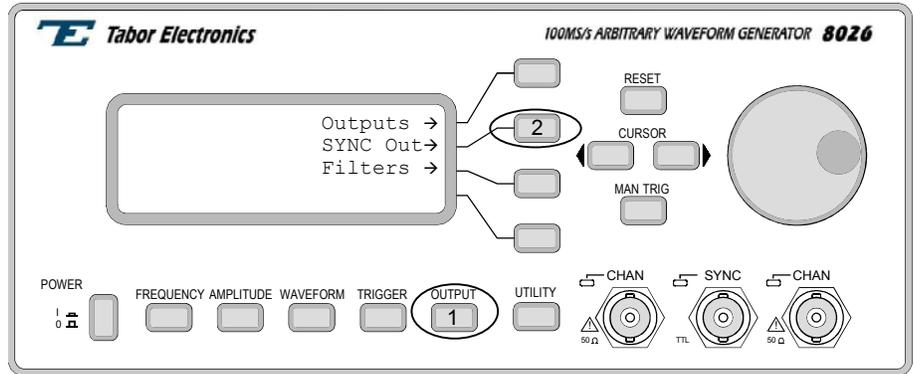
Figure 3-12. Selecting the SYNC Output Menu
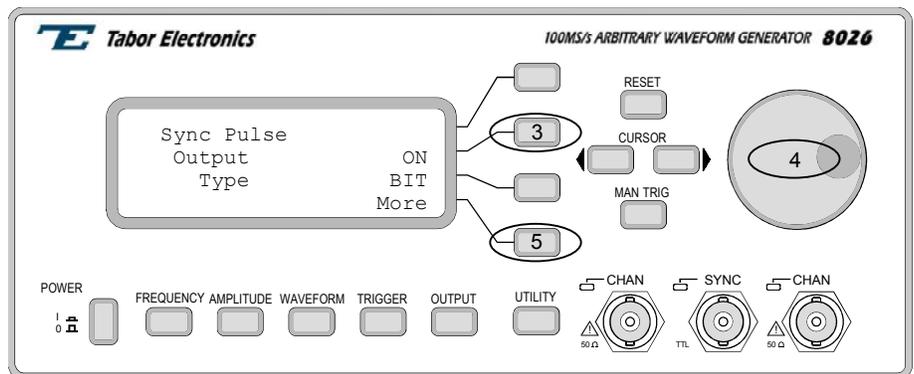
Figure 3-13. Enabling the SYNC Output
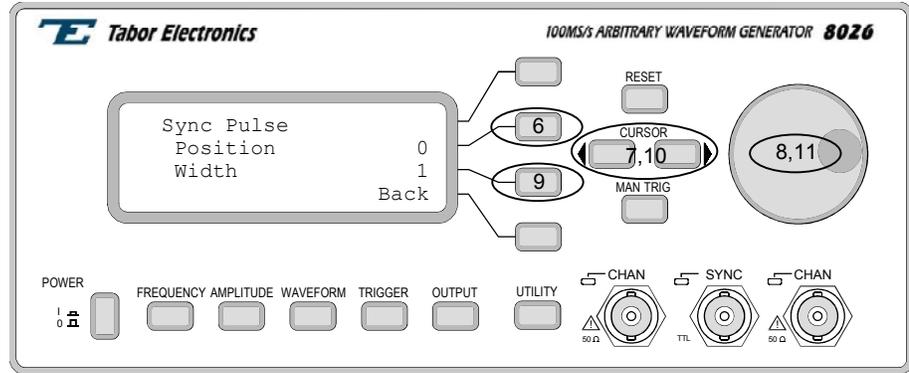
Figure 3-14. Modifying the SYNC Parameters

Selecting the Filter Type

Two filters are available for each channel. These filters have fixed cutoff frequencies of which their properties are specified in Appendix A. The built-in filters are switched in after the DAC circuit and are used for reducing the noise, harmonics and spurious signals above the cutoff frequency. More than 1 filter can be applied at any time, the effects being additive.

The built-in filters are available for the user in standard, arbitrary and sequenced modes. The only function where the Model 8026 does not allow external control is when standard sinusoidal waveform is selected.

Note

The default output function of the generator is the sine waveform. The instrument is using filters to generate this waveform therefore, the state of the filters can not be changed until another output function is selected. A setting conflict error will occur if one attempts to change the filter state before changing to another output function.

Use Figures 3-15 and 3-16, along with the following description to learn how to select filters. Before you change filters, make sure that your output waveform is different than sinewave.

1. Press the OUTPUT button
2. Select and press the Filters
3. Select and press either Filter CH1 or Filter CH2
4. Rotate the dial to select one of the filter options:

   NONE – No filter is selected

   25M – 25 MHz filter is selected

   50M – 50 MHz filter is selected

ALL – Both 25 MHz and 50 MHz are connected in series

If you do not plan on using the filters, make sure that you leave the selection NONE. This will eliminate confusing setting conflicts
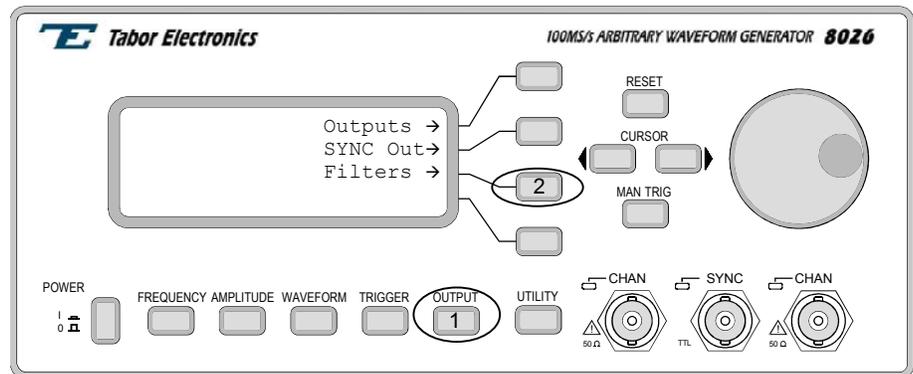
Figure 3-15. Selecting the Filters Menu
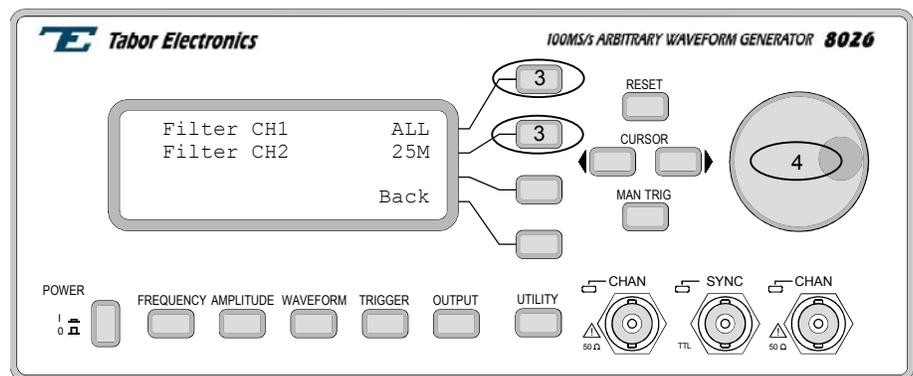
Figure 3-16. Selecting Filters

Selecting the SCLK Source and Reference

In cases where synchronization to other instrument in the system is needed, you may use the rear panel SCLK input for replacing the internal sample clock generators. The external sample clock input accepts ECL level signals, terminated to –2V into 50Ω.

The sample clock reference that is available with the 8026 is 1ppm as standard.
The External reference input is available for applications requiring better accuracy and stability than the internal reference. The rear panel EXT REF connector accepts 10MHz, TTL level signals only.

Use Figures 3-17 and 3-18, along with the following description to

learn how to select filters.

1. Press the UTILITY button
2. Select and press the SCLK Control option
3. Select and press either Sclk Source or Sclk Reference
4. Rotate the dial to select one of the source options:

   INT – Selects internal control

   EXT – Selects external control
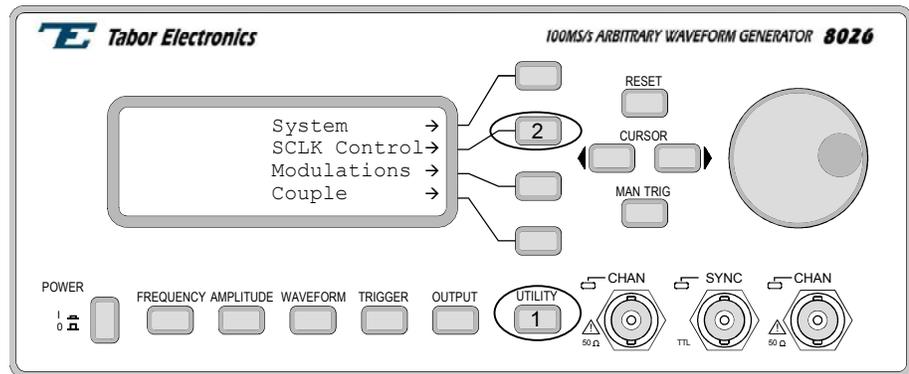
Figure 3-17. Selecting the SCLK Control Menu
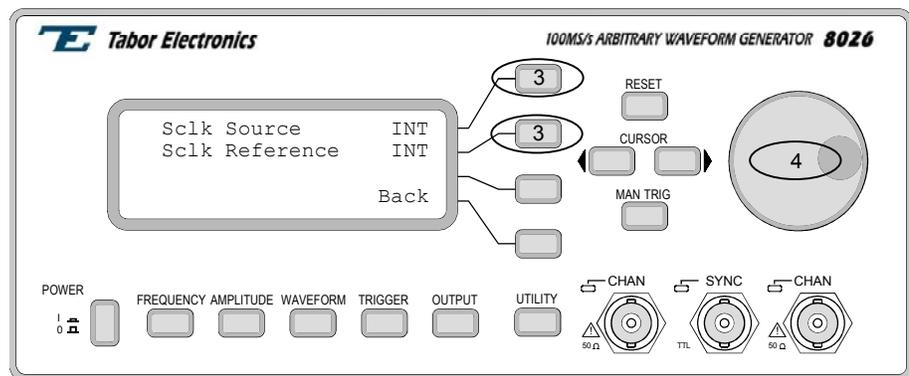
Figure 3-18. Selecting SCLK Reference Source

Modulating the Output

Utilizing DDS (direct digital synthesis) technology, the Model 8026 is extremely frequency-agile. Changing the sample clock frequency is as easy as changing bits at the DDS control. The DDS has an extremely wide dynamic range with excellent linearity throughout the complete range. The properties of the DDS are passed on directly to the output, therefore, the frequency is modulated within an extremely wide band, without loosing linearity. For example, the 8026 can sweep, using linear steps, from minimum to its maximum frequency whereas similar instruments with VCO design can sweep through 3 decades only.

The 8026 has the following modulation options: FM, FSK, Ramped FSK, Sweep and AM. These functions are described below.

FM

Frequency modulation is the most common modulation scheme used in telecommunications. In this mode, the sample clock frequency is modulated and placed at the rear-panel sine output. FM modulation is also available from the front panel outputs, except the frequency must be computed from the sample clock and the number of points that generate the output waveform. Confusing? Yes, a bit, but some examples below will remove your confusion.

The 8026 can be modulated using two different waveform types:

1. Modulation with built-in waveforms. Using this FM mode, you may select your modulating waveform from a library that has four pre-defined waveforms: Sine, Triangle, Square and Ramp. The manual refers to this mode as "Standard FM Mode"

2. Modulation with downloaded waveforms. To use this FM mode, you must generate coordinates for the modulating waveform with an external application, such as ArbExplorer[©], then download this waveform to the modulating waveform memory. This mode is referred to as "Arbitrary FM Mode" in this manual. The arbitrary waveform for this mode is separated from the main arbitrary memory of the 8026, is variable in length from 10 to 20000 points and has separate sample clock control. The sample clock for the arbitrary FM memory is controlled with 7 digits, from 1mS/s to 2 MS/s. The equivalent minimum and maximum frequencies are 50 nHz to 200 KHz. While the low frequency is very useful in applications like wander and slow drifting oscillators, the higher frequencies are as much needed for testing fast modems, PLL circuits and for wide-band digital modulation technology.

To generate FM you must specify four parameters:

1. Carrier Sample Clock – This is the frequency of the sample as

displayed after you press the FREQUENCY button. Note that the first line on the display shows Freq and some value and the second line shows Sclk and value. This Sclk is what you need to interpret your carrier frequency. This is also the frequency that is available as sine waveform at the rear-panel sine output. If you do not care to work with your sine output but want to generate FM from the front panel, then you must define your carrier frequency first. Here is what you do:

When generating standard waveforms, your sample clock frequency is computed automatically. In fact, when you program your waveform frequency, the Sclk value is updated automatically. You cannot access or modify the sample clock field, it is there for reference only. Although you cannot change the sample clock, this is the value you'll be using for the FM carrier.

It is a bit simpler when you generate arbitrary waveforms. The only value that you see on the display is Sclk and this is the value of your carrier frequency. It is also the frequency of the sine waveform at the rear-panel sine output connector. However, if you want to know the frequency of your signal at the front panel outputs, then you must computed it yourself from the sclk frequency, divided by the number of points in your active segment. For example, if your sclk is 100 MS/s and you have a 1000-points waveform, then your output frequency is 100 KHz.

2. Deviation Range – This parameter specifies the range of which the sample clock frequency will deviate from the carrier frequency setting. The deviation range is symmetrical about the carrier frequency. As was mentioned before, the 8026 has an extremely wide deviation range, without scarifying linearity. For example you can set 25 MS/s carrier with 40 MS/s deviation, that is, your carrier frequency will swing from 5 MS/s to 45 MS/s.

3. Modulating Waveform – Defines the shape and type of the modulating waveform. You can modulate the carrier with internal waveforms only. In other words, you cannot apply your own modulating carrier to an external input. The 8026 has a library of waveforms built into the software that allows selection of standard waveforms, such as, sine, triangle, square and ramp. As explained above, this mode is called "Standard FM Mode" You can also generate your own modulating waveform but, for this purpose, you must use external application such as ArbExplorer. Again, this mode is called "Arbitrary FM Mode".

4. Modulating Frequency – Defines the rate of the modulating frequency. If you use the standard FM mode, the maximum modulating frequency is 100 KHz. Using the arbitrary FM mode, the maximum envelope frequency is increased to 200 KHz.

Refer to Figures 3-19 and Figure 3-20 throughout the following description to learn how to program the 8026 to generate frequency

modulation.

Example 1, Modulating Standard Waveforms Using the Standard FM Mode

This example will show how to generate frequency modulation using the standard waveforms and the Standard FM Mode. The output frequency will be set to 1 MHz, Deviation Range to 100 KHz and 100 Hz sine as the modulating waveform. We'll monitor the frequency modulation from the front-panel outputs, then compare the results to the rear-panel sine output connector.

1. Press the FREQUENCY button. Modify Freq to display 1.000000 MHz. Observe and note that the Sclk parameter is showing 64.00000 MS/s. In standard waveform mode, the Sclk parameter is set automatically by the instrument and cannot be modified directly. The information that we need from this display is the number of points that the 8026 needs to generate 1 MHz. We can find this out from sclk/freq = 64/1 = 64 points. The number 64 will be needed later to compute the deviation range parameter.

2. Press the UTILITY button

3. Select and press the Modulations option

4. Select and press the FM option

5. Select the FM State option

6. Rotate the dial clockwise to turn the FM function ON

Select and press the STD. FM Type option. Note that the default modulating waveform is SIN (sinewave). Leave this option as is for now. Later you can practice modifying the modulating waveform to other shapes.
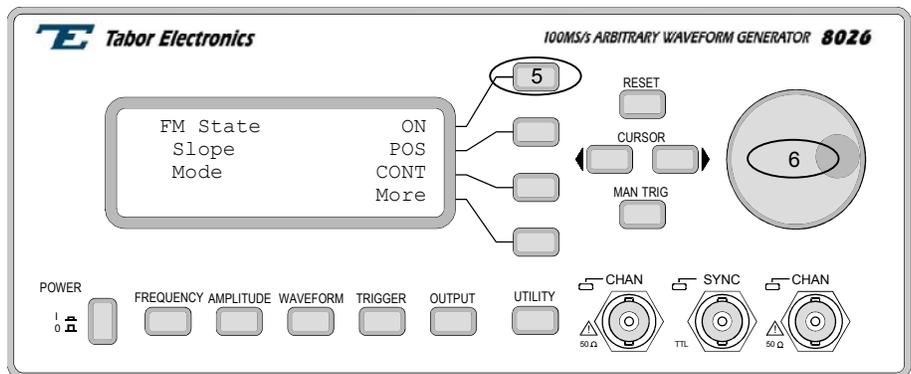
Figure 3-19. Selecting FM Type

7. Select the Dev parameter. This parameter sets the deviation range value. Remember that we computed the number of points that the 8026 is using for generating 1 MHz? We need to use this number to program the deviation range. We want the front panel output to deviate by 100 KHz, 50 KHz above and 50 KHz below the carrier frequency. Since the deviation range is

controlling sample clock, the value that you have to put on the display is 100 KHz x 64 = 6.4 MS/s

8. Use the cursor to point to the digit you want to change and rotate the dial to adjust the display to 6.4 MS/s

9. Select the Freq parameter. This parameter sets the modulating frequency. There is no need to compute and convert this parameter because the frequency is given in units of Hz.

10. Use the cursor to point to the digit you want to change and rotate the dial to adjust the modulating frequency to 100 Hz

11. You may press More to access triggered FM modes, or set marker position
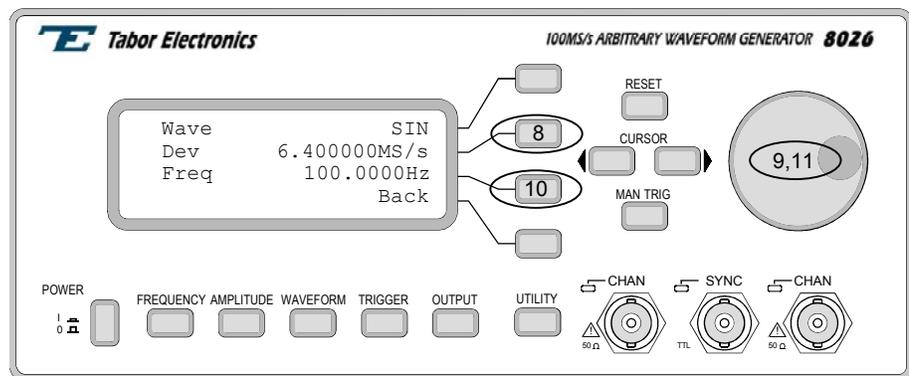
Figure 3-20. Programming Standard FM Mode Parameters

If you did not make any programming errors, the front panel outputs will generate frequency modulated waveforms having carrier frequency of 1 MHz, deviation frequency of 100 KHz and modulating frequency of 100 Hz.

We'll now check the results at the rear-panel sine output connector and compare what we get there to what we see on the front panel. Remove the cable from the main output connector and connect to the rear-panel SINE OUT connector. Here is what you should expect to see when you check this output:

1. Carrier is sine waveform with fixed amplitude level of 1V into 50Ω.

2. Carrier frequency is 64 MHz

3. Deviation range is 3.2 MHz above and 3.2 MHz below the carrier

4. Modulating waveform is sine waveform and modulating frequency is 100 Hz.

NOTE

If you selected Triggered mode for the 8026, you will not be able to turn the FM function on. There are some trigger modes that could be accessed through the FM menu; These modes behave differently then the main operating modes. For example, triggered FM means that the output is actually in continuous mode but, every time you trigger the 8026, a single FM cycle is initiated hence the carrier frequency sweeps though the deviation range and then idles at the carrier node.

Example 2, Modulating Standard Waveforms Using the Arbitrary FM Mode

This example will show how to generate frequency modulation using the standard waveforms and the Arbitrary FM Mode. The output frequency will be set to 2 MHz, Deviation Range to 100 KHz and 100 Hz sine as the modulating waveform. We'll monitor the frequency modulation from the front-panel outputs, then compare the results to the rear-panel sine output connector.

1. Press the FREQUENCY button. Modify Freq to display 2.000000MHz. Observe and note that the Sclk parameter is showing 64.00000MS/s. In standard waveform mode, the Sclk parameter is set automatically by the instrument and can not be modified directly. The information that we need from this display is the number of points that the 8026 needs to generate 2 MHz. We can find this out from sclk/freq = 64/2 = 32 points. The number 32 will be needed later to compute the deviation range parameter.

2. Press the UTILITY button

3. Select and press the Modulations option

4. Select and press the FM option

5. Select the FM State option

6. Rotate the dial clockwise to turn the FM function ON

7. Select and press the ARB FM Type option. Note that only parameter you can access is the FM Sclk. This parameter is used for clocking the waveform memory that is used specifically for the Arbitrary FM Mode. This sample clock parameter has no effect on the main sample clock. The default value is 1 MS/s. Note this value because it will be used for calculating the frequency of the arbitrary wave that will be used as the modulating waveform
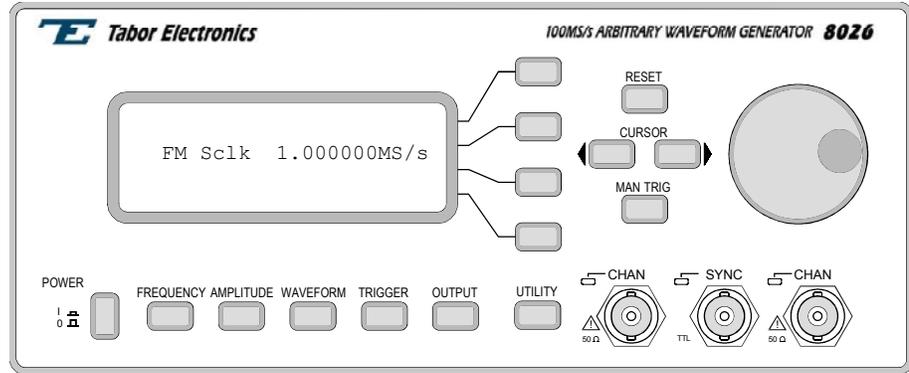
Figure 3-21. Programming the Sample Clock for the Modulating Arbitrary Waveform

8. You'll now need to use ArbExplorer to generate your modulating waveform. Invoke ArbExplorer, select the modulated waveforms option and press the FM Composer button on the Modulation panel.

9. Refer to the Figure 3-22 and modify the Wavelength field to have 10,000 points. After you modify the field, press the Wavelength button. The number of points is derived from the following relationship:

Number of Points = FM Sclk / 100Hz = 1M / 100 = 10,000

10. Now you have to program the deviation range. At the beginning of this example we computed and showed that the 8026 is using 32 points to generate this waveform. Note that the deviation range is given on sample per second and use the following relationship to compute the number for the deviation range:

Deviation Range = 100 KHz x 32 = 3.2 MS

After you complete updating the above field, the FM Composer will have the following coordinates:

The Y axis shows the frequency deviation range. The center line shows 64 MS and the high and low limits are +65.6 MS and –62.4 MS, respectively.

The X axis shows the length of the arbitrary modulating waveform. The length is in points. In our example, the first point is 0 and the last is 9999 (total of 10,000 points).

11. Now that you have the Y and X parameters set, you can generate the shape of the modulating waveform. ArbExplorer has a library of standard waveform that include: Sine, Triangle, Square, Exponential, Pulse and Random Noise waveforms. You can use one of these waveforms, or import coordinates from an

external file to generate more complex waveforms. We'll use a simple sine waveform for this example. From the Waves menu select the Sine. Leave the parameters in the Sine dialog box as is and press OK.

12. The last step is to download the modulating waveform to the 8026. Press the download button to send the modulating waveform to the instrument.
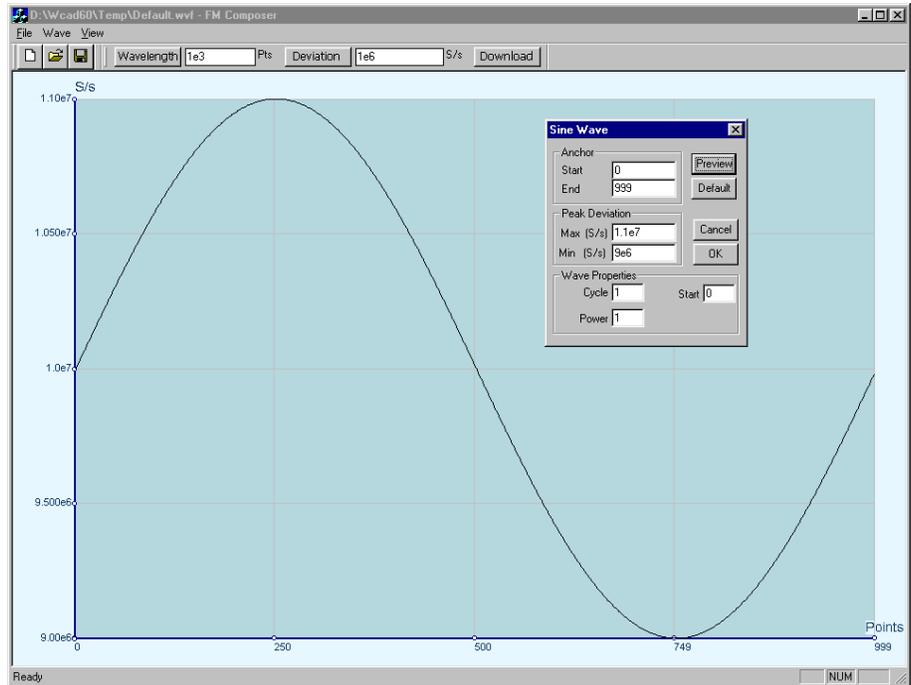
Figure 3-22. Using ArbExplorer to Generate Arbitrary Modulating Waveforms

If you did not make any programming errors, the front panel outputs will generate frequency modulated waveforms having carrier frequency of 2 MHz, deviation frequency of 100 KHz and modulating frequency of 100 Hz.

We'll now check the results at the rear-panel sine output connector and compare what we get there to what we see on the front panel. Remove the cable from the main output connector and connect to the rear-panel SINE OUT connector. Here is what you should expect to see when you check this output:

1. Carrier is sine waveform with fixed amplitude level of 1V into 50Ω.

2. Carrier frequency is 64 MHz

3. Deviation range is 3.2 MHz above and below the carrier

4. Modulating waveform is sine waveform and modulating frequency is 100 Hz.

NOTES

1. If you selected Triggered mode for the 8026, you will not be able to turn the FM function on. There are some trigger modes that could be accessed through the FM menu; These modes behave differently then the main operating modes. For example, triggered FM means that the output is actually in continuous mode but, every time you trigger the 8026, a single FM cycle is initiated hence the carrier frequency sweeps though the deviation range and then idles at the carrier node.

Frequency Modulating Arbitrary Waveforms

In Examples 1 and 2 above, we showed you how to frequency modulate standard waveforms. You can use these examples as guideline for modulating arbitrary waveforms. Generating arbitrary waveforms is discussed later in this chapter, so before you try modulating such waveforms, make sure that you first familiarize yourself with the arbitrary waveform generation techniques. In general, arbitrary waveforms are made of waveform data and sample clock. The rate of the sample clock determines the frequency of the output waveform. For example, if your sample clock is 1MS/s and your waveform segment is 1000 points long, the frequency of the waveform is 1M / 1000 = 1 KHz.

The above example is somewhat oversimplified. What is important to remember is that regardless if you are generating standard or arbitrary waveforms, the FM function modulates the sample clock and the parameters that you program for FM affects the SINE OUT signal directly while the frequency characteristics for the front-panel outputs must be computed.

FSK

In Frequency Shift Keying (FSK), the outputs of the 8026 hop between two sample clock settings. You program the first sample clock setting in the FREQUENCY menu and the shifted sample clock frequency in the UTILITY menu. After you program the carrier and the shifted frequencies, the instrument is ready to accept hop commands. Hop control is done from the rear-panel FSK IN. This input is level sensitive. When the FSK IN signal is low, the output generates carrier sample clock, changing the level to high modifies the output to shifted sample clock.

FSK is executed on the sample clock frequency. If you are just using sine waveforms, then we recommend that you use the rear-panel SINE OUT waveform. Using this output, the sample clock and the shifted sample clock frequencies are used as nominal values. If you want to use the FSK function from the front panel, then you

must compute the actual carrier and shifted frequencies in a similar way as was described above for the FM function.

To generate FSK you must specify two parameters:

1. Carrier Sample Clock – This is the frequency of the sample as displayed after you press the FREQUENCY button. Note that the first line on the display shows Freq and some value and the second line shows Sclk and value. This Sclk is what you need to interpret your carrier frequency. This is also the frequency that is available as sine waveform at the rear-panel sine output. If you do not care to work with your sine output but want to generate FSK from the front panel, then you must define your carrier frequency first. Here is what you do:

When generating standard waveforms, your sample clock frequency is computed automatically. In fact, when you program your waveform frequency, the Sclk value is updated automatically. You can not access or modify the sample clock field, it is there for reference only. Although you can not change the sample clock, this is the value you'll be using for the FSK carrier.

It is a bit simpler when you generate arbitrary waveforms. The only value that you see on the display is Sclk and this is the value of your carrier frequency. It is also the frequency of the sine waveform at the rear-panel sine output connector. However, if you want to know the frequency of your signal at the front panel outputs, then you must computed it yourself from the sclk frequency, divided by the number of points in your active segment. For example, if your sclk is 100MS/s and you have a 1000-points waveform, then your output frequency is 100 KHz.

2. Shifted Frequency – Defines the shifted frequency value. The 8026 will hop to the shifted frequency when commanded to do so. Note that the instrument changes the frequencies at the SINE OUT connector coherently, that is, after the shift command is accepted, the generator completes the waveform to its last point or phase and only then the next frequency is "kicking in". This way you can be sure that discontinuities in the output waveforms is not disturbed.

While using the FSK IN as hop control, the instrument will shift on positive-going transitions only. Note that the FSK input is level sensitive, so, although the frequency shifts on transitions, the output remains at the shift frequency as long as the level remains true. The output will remain at carrier frequency as long as the FSK input remains TTL low (false) and at shifted frequency as long as the output remains TTL high (true).

Refer to Figures 3-23 throughout the following description to learn how to program the 8026 to generate FSK modulation.

Example 1, FSK Using Standard Waveforms

This example will show how to generate FSK modulation using the standard waveforms. The carrier frequency will be set to 1 MHz and the shifted frequency to 1.5 MHz. We'll monitor the FSK modulation from the front-panel outputs, then compare the results to the rear-panel sine output connector.

1. Press the FREQUENCY button. Modify Freq to display 1.000000MHz. Observe and note that the Sclk parameter is showing 64.00000MS/s. In standard waveform mode, the Sclk parameter is set automatically by the instrument and can not be modified directly. The information that we need from this display is the Sclk value and the number of points used for generating the current waveform. In this case, the number of points is computed from the relationship

$$N = Sclk/Freq = 64MS/1MHz = 64 \text{ points.}$$

2. Now it becomes a bit tricky. When generating standard waveforms, the Model 8026, depending on the output frequency, automatically sets the number of points and their associated sample clock. In FSK mode, only the sample clock hops between frequencies, but the number of points remain constant. This means that 64 waveform points will remain for the carrier and for the shifted waveforms. From this statement we can now compute the sample clock for the shifted frequency using the following equation:

$$F = 64 \text{ Pts x } 1.5 \text{ MHz} = 96MS/s.$$

3. Press the UTILITY button

4. Select and press the Modulations option

5. Select and press the FSK option

6. Select the FSK State option

7. Rotate the dial clockwise to turn the FSK function ON

8. Select and press the Sclk2 parameter

9. Rotate the dial to adjust the Sclk2 parameter to 96 MS/s. This value was selected using step 2 above.

10. Apply your FSK control signal to the trigger input
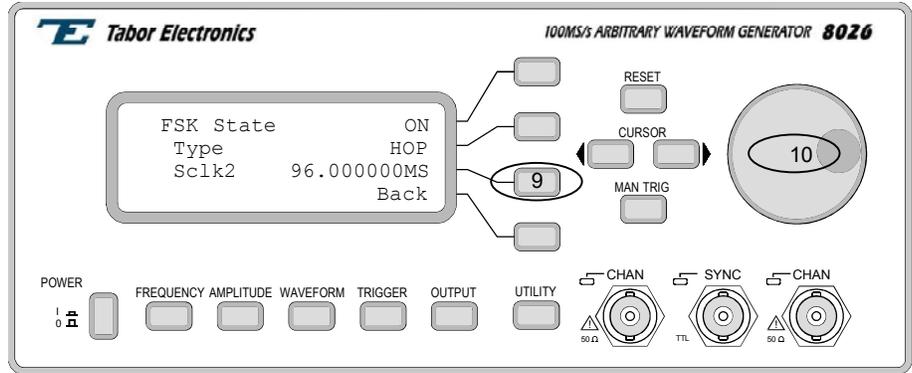
Figure 3-23. Adjusting FSK Parameters

If you did not make any programming errors, the front panel outputs will generate FSK modulated waveforms having carrier frequency of 1 MHz and shifted frequency of 1.5 MHz.

We'll now check the results at the rear-panel sine output connector and compare what we get there to what we see on the front panel. Remove the cable from the main output connector and connect to the rear-panel SINE OUT connector. Here is what you should expect to see when you check this output:

1. Carrier is sine waveform with fixed amplitude level of 1V into 50Ω.

2. Carrier frequency is 64 MHz

3. Shifted frequency is 96 MHz

Example 2, FSK Using Arbitrary Waveforms

This example will show how to generate FSK modulation using the arbitrary waveforms. The carrier frequency will be set to 10 KHz and the shifted frequency to 25 KHz. We'll monitor the FSK modulation from the front-panel outputs, then compare the results to the rear-panel sine output connector.

Before we start with our 8026 setting, we must know the length of the arbitrary segment that we'll use. To simplify matters, lets download a triangular waveform to the segment that was constructed using 1000 waveform points. From this point it is easy to compute the frequency and the sample clocks that we need for the above example.

We want the carrier frequency to be 10 KHz. Bearing in mind that the output frequency is equal to the sample clock frequency divided by the number of points, the sample clock must be programmed to be 10 MS/s (10M / 1000 = 10 KHz). Using the same method, the sample clock for the shift frequency is 25 MS/s (25M / 1000 = 25 KHz).

Now that we have both frequency values for the carrier and the shifted sample clock, we can go directly to front panel programming. Refer to Figure 3-23 for this example

1. Press the FREQUENCY button. Modify Sclk display 10.00000MS.
2. Press the UTILITY button
3. Select and press the Modulations option
4. Select and press the FSK option
5. Select the FSK State option
6. Rotate the dial clockwise to turn the FSK function ON
7. Select and press the Sclk2 parameter
8. Rotate the dial to adjust the Sclk2 parameter to 25 MS
9. Apply your FSK control signal to the trigger input

If you did not make any programming errors, the front panel outputs will generate FSK modulated waveforms having carrier frequency of 10 KHz and shifted frequency of 25 KHz.

We'll now check the results at the rear-panel sine output connector and compare what we get there to what we see on the front panel. Remove the cable from the main output connector and connect to the rear-panel SINE OUT connector. Here is what you should expect to see when you check this output:

1. Carrier is sine waveform with fixed amplitude level of 1V into 50Ω.
2. Carrier frequency is 10 MHz
3. Shifted frequency is 25 MHz

Ramped FSK

Ramped FSK is very similar to the FSK function except, when the hop command is received, the output frequency ramps to the shifted frequency at a rate set by the ramp time generator. The ramp time is programmable from 10 μs to 1 second.

The procedure how to set up the carrier and shifted frequencies for the ramped FSK function is exactly as shown above for the Hopped FSK except an additional parameter has to be specified – ramp time.

Do the following to access and modify the ramp time parameter:

1. Press the UTILITY button
2. Select and press the Modulations option
3. Select and press the FSK option
4. Select the press the FSK Type option
5. Rotate the dial clockwise to modify the FSK type to RAMP
6. Press again the FSK Type button

7. Using the dial, program the FSK Ramp Time to the desired value

---

NOTE

Shifted frequency value in Ramped FSK mode must always be higher than the carrier frequency. Reverse setting will not allow the 8026 to execute FSK operation. Also, SCLK2 - SCLK1 ≥10KHz

---

Sweep

In sweep modulation, the 8026 sweeps from start frequency set by the Sclk parameter to stop frequency set by Sweep Stop parameter. Both the sweep start and the sweep stop control the sample clock generator at the rear-panel sine output. Sweep modulation is also available from the front panel outputs, except the frequency must be computed from the start sample clock and the number of points that generate the start output waveform. Confusing? Yes, a bit, but some examples below will remove your confusion.

To generate Sweep you must specify five parameters:

1. Start Sample Clock – This is the frequency of the sample clock as displayed after you press the FREQUENCY button. Note that the first line on the display shows Freq and some value and the second line shows Sclk and value. This Sclk is what you need to interpret your sweep start frequency. This is also the frequency that is available as sine waveform at the rear-panel sine output. If you do not care to work with your sine output but want to generate sweep from the front panel, then you must define your sweep start frequency first. Here is what you do:

When generating standard waveforms, your sample clock frequency is computed automatically. In fact, when you program your waveform frequency, the Sclk value is updated automatically. You can not access or modify the sample clock field, it is there for reference only. Although you can not change the sample clock, this is the value you'll be using for the sweep stop frequency.

It is a bit simpler when you generate arbitrary waveforms. The only value that you see on the display is Sclk and this is the value of your sweep start frequency. It is also the frequency of the sine waveform at the rear-panel sine output connector. However, if you want to know the frequency of your signal at the front panel outputs, then you must compute it yourself from the sclk frequency, divided by the number of points in your active segment. For example, if your sclk is 100MS/s and you have a 1000-points waveform, then your output frequency is 100 KHz.

2. Stop Sample Clock – This parameter specifies where the sweep will stop. This parameter is available from within the Sweep Modulation menu. The sweep stop programs the sample clock in units of sample per second. Note that sweep stop does not necessarily have to have lower value than the sweep stop frequency. If you define sweep stop frequency higher than the sweep start frequency, then the generator will sweep up. You can also reverse the direction of the sweep by using the sweep direction parameter.

HINTS

In standard waveform mode, the Sclk parameter and the number of waveform points are set automatically by the instrument and can not be modified directly. During sweep modulation, only the sample clock is changing but the number of waveform points remain constant. Therefore, computing the number of waveform points used by the instrument is critical to the sweep operation. For example, if you set your start frequency at 50 MHz, the 8026 can use only two waveform points. While the instrument can generate a fairly nice sine waveform at 50 MHz by using just two points (and special filters), the same two points will definitely look like square waveforms if the sample clock is reduced below 10 MS/s.

As a reverse example, if you sweep modulate the output from a start frequency of 150 KHz, the generator automatically sets the Sclk to 76.8 MS/s and the number of waveform points to 512. If you try to sweep up from this point, then your maximum frequency will be 195,312 KHz! (100MS/s / 512 Pts = 195,312 KHz). Therefore and due to these limitations, it is recommended using the sweep modulation in conjunction with arbitrary waveforms because you are in complete control of the number of points and the sample clock frequency you are using to generate your waveform.

It is also recommended to use the SINE OUT at the rear panel because the start and stop frequency setting are available at this output at nominal values, without needing to compute values for different waveform types.

3. Sweep Type – Defines the steps of which the sample clock increments, or decrements from start to stop frequencies. A choice is provided between linear and logarithmic step. If you select linear sweep the frequency range is divided to 1000 linear increments and the sample clock hops through these frequency-increments within a

time interval which is set by the sweep time parameter. Likewise, using the logarithmic sweep type, the frequency span between the start and stop frequencies is divided by 1000 logarithmic steps.

4. Sweep Direction – Defines the direction of the sweep. UP sets sweep direction from start frequency to stop frequency; DOWN reverses the sweep direction so the output sweeps from stop frequency to start frequency.

5. Sweep Time – Defines the time interval that it takes for the 8026 output to execute one complete sweep. The sweep time is generated with an asynchronous free-running time interval generator that has programmable intervals from 1 ms to 1000 seconds.

Refer to Figures 3-24 and Figure 3-25 throughout the following description to learn how to program the 8026 to generate sweep modulation.

Example 1, Generating Sweep Using Standard Waveforms

This example will show how to generate sweep modulation using the standard waveforms. The start frequency will be set to 100 Hz and the stop frequency to 25 KHz. We'll be using linear sweep in the down direction in 10 ms. We'll monitor the sweep modulation from the front-panel outputs, then compare the results to the rear-panel sine output connector.

1. Press the FREQUENCY button. Modify Freq to display 100.0000Hz. Observe and note that the Sclk parameter is showing 102.4000KS/s. In standard waveform mode, the Sclk parameter is set automatically by the instrument and can not be modified directly. The information that we need from this display is the Sclk value and the number of points used for generating the current waveform. In this case, the number of points is computed from the relationship

   $$N = Sclk/Freq = 102.4KS/s / 100Hz = 1024 \text{ points.}$$

2. Now it is time to check if the number of points will not conflict at the stop frequency. We can check this by dividing the maximum possible sample frequency by the number of points from the above calculations.

   $$Fstop\_max = 100MS/s / 1024Pts = 97.65625KHz$$

   In this case we should be safe because we plan to sweep to 25KHz only.

3. Now we have to compute the sample clock frequency at the stop point. Knowing already the number of points we have in this waveform, the sample frequency at the stop point is computed using the following equation:

   $$Fstop = 1024Pts \times 25KHz = 25.6MS/s.$$

4. Press the UTILITY button

5. Select and press the Modulations option

6. Select and press the Sweep option

7. Select the Sweep State option

8. Rotate the dial clockwise to turn the Sweep function ON

9. Select and press the Sweep Time parameter

10. Move the cursor below the digit you want to change and rotate the dial to adjust the Sweep Time parameter to 10ms.

11. Select and press the Stop parameter

12. Move the cursor below the digit you want to change and rotate the dial to adjust the Sweep Stop parameter to 25.6MS/s

13. Press More to program the rest of the sweep parameters

14. Select and press the Direction parameter

15. Rotate the dial clockwise to modify the direction to DOWN
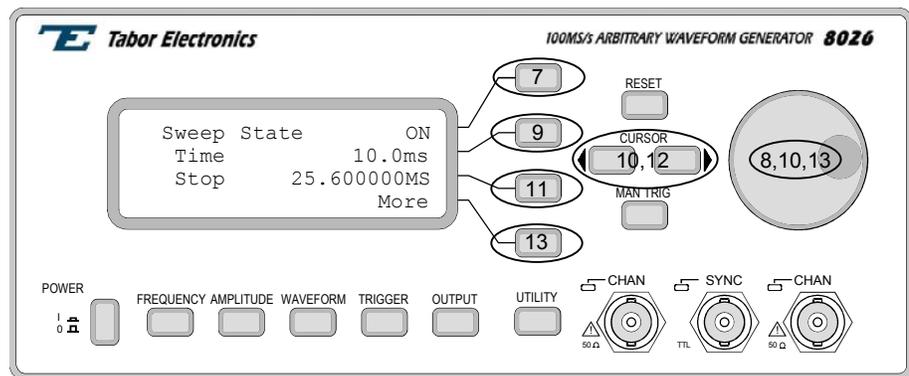
Figure 3-24. Adjusting Sweep Parameters
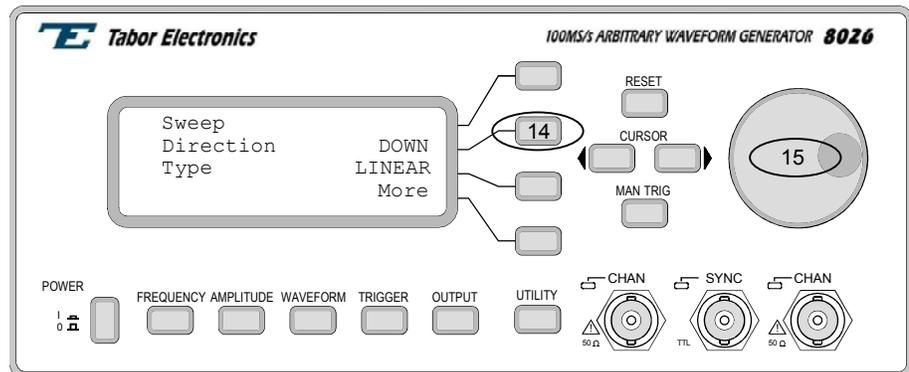
Figure 3-25. Adjusting Sweep Parameters (continued)

If you did not make any programming errors, the front panel outputs will generate linearly swept waveforms from 25 KHz to 100 Hz in 10

ms.

We'll now check the results at the rear-panel sine output connector and compare what we get there to what we see on the front panel. Remove the cable from the main output connector and connect to the rear-panel SINE OUT connector. Here is what you should expect to see when you check this output:

1.   Waveform is sinewave having fixed amplitude level of 1V into 50Ω.

2.   Sine waveform sweep from 25.6 MHz to 102.4 KHz in 10 ms. Sweep step is linear.

Example 2, Generating Sweep Using Arbitrary Waveforms

This example will show how to generate sweep modulation using the arbitrary waveforms. The start frequency will be set to 100 Hz and the stop frequency to 25 KHz. We'll be using linear sweep in the up direction in 10 ms. We'll monitor the sweep modulation from the front-panel outputs, then compare the results to the rear-panel sine output connector.

Before we start with our 8026 setting, we must know the length of the arbitrary segment that we'll use. To simplify matters, let us download a triangular waveform to the segment that was constructed using 100 waveform points. From this point it is easy to compute the frequency and the sample clocks that we need for the above example.

We want the start frequency to be 100 Hz. Bearing in mind that the output frequency is equal to the sample clock frequency divided by the number of points, the start sample clock must be programmed to be 10 KS/s (10K / 100 = 100 Hz).

1.   Press the FREQUENCY button. Modify Sclk to display 10.00000KS/s.

2.   Now it is time to check if the number of points will not conflict at the stop frequency. We can check this by dividing the maximum possible sample frequency by the number of points from the above calculations.

Fstop_max = 100MS/s / 100Pts = 1MHz

In this case we should be safe because we plan to sweep to 25KHz only.

3.   Now we have to compute the sample clock frequency at the stop point. Knowing already the number of points we have in this waveform, the sample clock frequency at the stop point is computed using the following equation:

Fstop = 100Pts x 25KHz = 2.5MS/s.

4.   Press the UTILITY button

5.   Select and press the Modulations option

6.   Select and press the Sweep option

7. Select the Sweep State option

8. Rotate the dial clockwise to turn the Sweep function ON

9. Select and press the Sweep Time parameter

10. Move the cursor below the digit you want to change and rotate the dial to adjust the Sweep Time parameter to 10ms.

11. Select and press the Stop parameter

12. Move the cursor below the digit you want to change and rotate the dial to adjust the Sweep Stop parameter to 2.5MS/s

If you did not make any programming errors, the front panel outputs will generate linearly swept waveforms from 100 Hz to 25 KHz in 10 ms.

We'll now check the results at the rear-panel sine output connector and compare what we get there to what we see on the front panel. Remove the cable from the main output connector and connect to the rear-panel SINE OUT connector. Here is what you should expect to see when you check this output:

1. Waveform is sinewave having fixed amplitude level of 1V into 50Ω.

2. Sine waveform sweeps from 10 KHz to 2.5 MHz in 10 ms. Sweep step is linear.

AM

Amplitude modulation, unlike frequency modulation is not digital. Modulating waveform must be applied to a rear-panel connector to modulate the 8026 outputs. Amplitude modulation control is separate for each channel, so you can amplitude modulate one channel without affecting the other.

Before using the AM function, you should get yourself familiar with some of the terms:

Carrier – This term define the signal and its properties as are available at the output connectors and without external intervention. The carrier signal could be any type and shape that the 8026 can generate.

Carrier Frequency – This term defines the frequency of the carrier waveform. In standard waveforms mode, the frequency of the waveform is displayed after you press the FREQUENCY button. In arbitrary waveforms mode, the frequency of the output waveform is derived from the sample clock frequency and the number of points used to generate the active waveform segment.

Envelope Waveform – The carrier signal is modulated with an external signal. This signal is referred to as the envelope waveform. This waveform is applied to the AM IN connector and must have specific amplitude to modulate the 8026 correctly

Envelope Frequency – This term defines the frequency of the

modulating waveform. The envelope frequency is bandwidth limited, so make sure you are within the specified limits before you apply this signal to the AM input connector.

AM Depth or Modulation Index – This term defines how far the external generator can affect the main output signal.

The 8026 can be amplitude modulated from an external signal only. This means that besides the carrier frequency, all other parameters are supplied externally by the user. Therefore, make sure that the signal that you apply to the AM input complies with the specification requirements set forth in Appendix A.

Use the following procedure to turn on the AM function:

1. Press the UTILITY button
2. Select and press the Modulations option
3. Select and press the AM option
4. Select the press AM State CH1, or AM State CH2
5. Rotate the dial clockwise to turn the AM function ON

Connect the modulating signal to the rear-panel AM IN connector and monitor the modulated waveform from the main outputs. Note that AM has no effect on the SINE OUT signal.

Adjusting Phase Offset Between Channels

The 8026 has two output channels that can generate various and numerous waveforms. Although the control over waveform parameters is separate for each channel, the sample clock is derived from a single source. Having a single source for both channels is of great advantage because of two main reasons: 1) There is no jitter between the two channels and 2) If we ignore the initial skew, both waveforms start at exactly the same phase.

Understanding the initial skew term is very important. If you set both channels to output square waveforms and then connect these signals to an oscilloscope. If you then set the oscilloscope to its fastest time base setting, you'll see the two rising edges of the 8026 signals. They do not overlap because the instrument has a skew spec of ±2 ns. Skew is caused as a result of many factors. Although the two channels were designed exactly the same, small variations in printed circuit board layout or component values are enough to cause skew. These factors were known during the design phase and were minimized as practical. On the other hand, skew can also be generated from external factors that are controlled by the user alone. Examples for these factors are variation in cable length and quality, as well as, non-symmetrical end termination. Therefore, if you want to eliminate skew between channels, you have to use exactly the same cable type, the same cable length and the same termination for the two channels.

There are times, however, that you do need to offset phase between channels. In that case, the 8026 lets you adjust phase-offset variations to a resolution of one point. When you do, just keep in mind that the initial skew will escort your programmed phase offset throughout the entire phase offset range.

Notice that phase offset resolution is given in points, not degrees. This is because the 8026 can generate arbitrary waveforms that you download from an external application. After you download waveforms, there is no way for the instrument to determine phase because all it has is just number of points and no further knowledge of the shape of your waveform. If you insist, you can convert points to degrees on your waveform from the following equation:

Phase (in °) = Phase point x 360 / number of waveform points

If you are using standard waveforms, you can determine the number of points that the 8026 is using from the FREQUENCY menu. Note the Freq and the Sclk values. The number of points is derived from the following relationship:

Number of Waveform Points = Sclk / Freq

After you know the number of points you can easily calculate the

phase point from the above equation.

When you modify the phase offset parameter, you are offsetting the phase of channel 2in reference to channel 1edge. For example, if you program phase offset of 100 points and check the results on an oscilloscope, what you should expect to see is channel 2start phase lagging channel 1by 100 points.

Phase offset range is limited to 999,999 points, so if you have the basic 8026 memory (1Meg) then you can phase offset throughout the entire range of your memory.

Refer to Figure 3-26 and to the following description to find out how to program phase offset between channels.
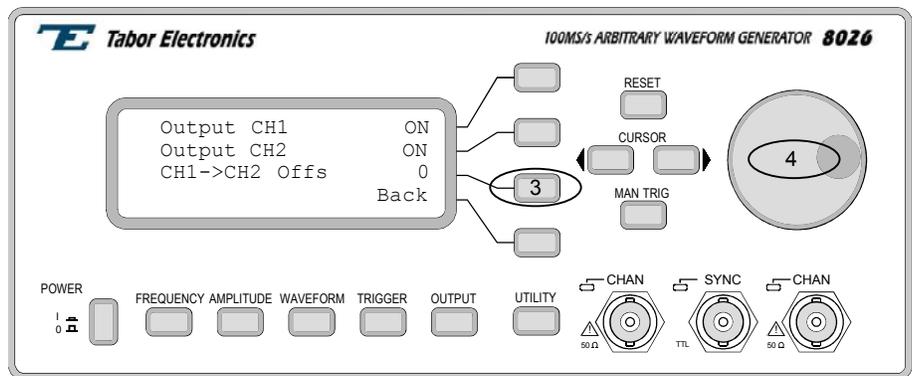
Figure 3-26. Adjusting Phase Offset Between Channels

1. Press the OUTPUT button
2. Select and press the Outputs option
3. Select and press CH1->CH2 Offs parameter
4. Rotate the dial adjust the phase offset parameter
5. You may also use the cursor buttons to move the pointer under the digit you want to modify

Synchronizing Multiple Instruments

The LW100 series comes in two versions: 8025 with a single output channel and 8026 with two output channels. In applications requiring more output channels, as well as, complete control over inter-channel synchronization, you can not just use separate instruments because the output signals will not be synchronized. The LW100 series has a special master/slave synchronization feature that allows connection of multiple instruments in a daisy chain. Not only that the channels are derived from the same master sample clock source but other signals are distributed from the master instrument, which enable tight control over start phase and jitter.

Connecting multiple instruments in master/slave configuration is extremely easy. All you need is two types of cables, the first being a standard BNC to BNC coax cable to feed the sample clock output and a 9-pin to 9-pin cable to feed the master-to-slave, or slave-to-slave controls. You can get the standard BNC cable from any store however, the master/slave cable must be bought from your Tabor distributor. Figure 3-27 shows the connection between master and slave instruments. Additional slave units can be daisy-chained in a similar way. What you have to remember is that SCLK OUT feeds an adjacent SCLK IN. Also, always have the last slave unit in the chain terminated with the special 9-pin terminator.
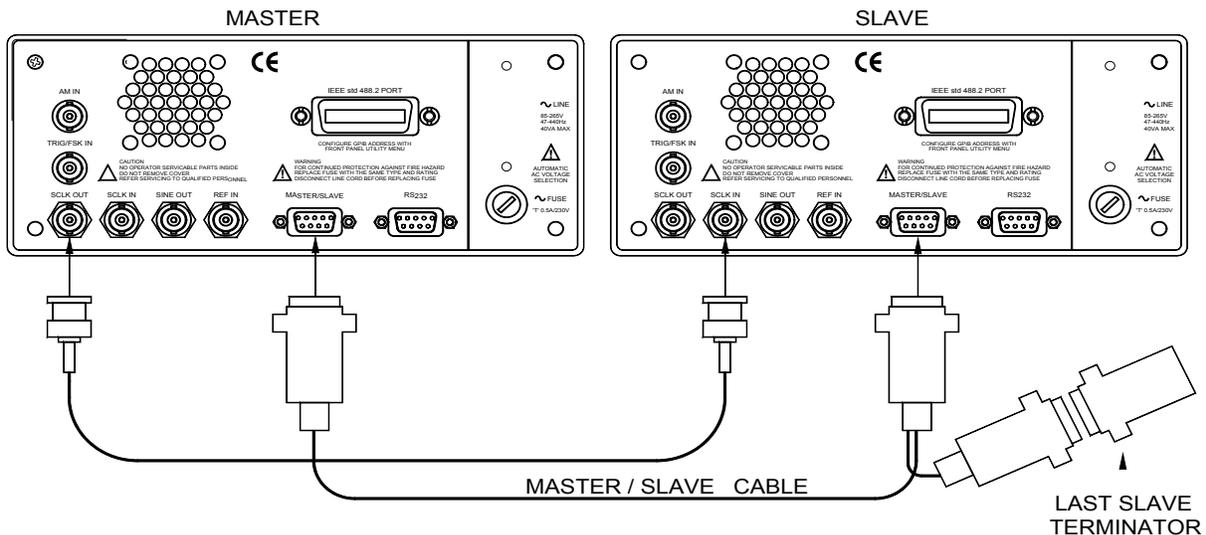
Figure 3-27. Wiring Diagram Master to Slave

Hard wiring the instruments from the back is not enough. You now have to set up the first instrument in the chain as master and the other instruments as slaves. Refer to Figure 3-28 and the following description to learn how to set 8026 instruments as master and slaves.

1. Connect your instruments as shown in Figure 3-27. Make sure you designate the first instrument in the chain as master. Also make sure the last slave is terminated with the 9-pin terminator.

2. On each of the instruments press the UTILITY button

3. Select and press the Couple button on all units

4. *On the slave units only*, select and press the Couple Mode button

5. On the slave units only, rotate the dial to select the Slave mode

6. On the master unit only, select and press the Couple State button

7. Finally, on the master unit only, rotate the dial to turn the Couple State ON
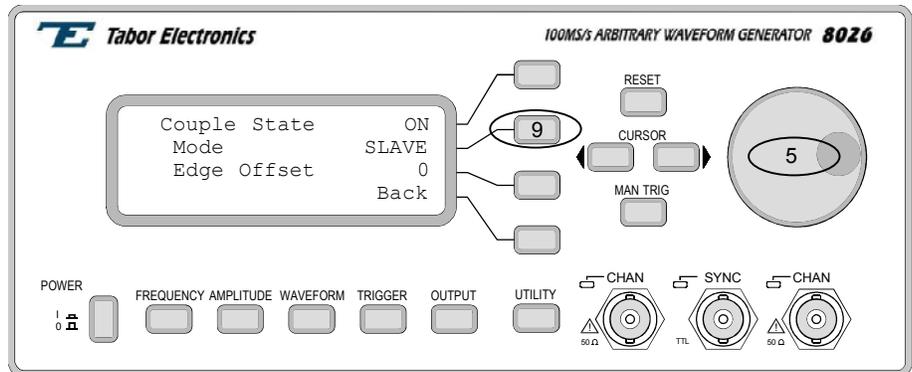
Figure 3-28. Setting Master/Slave Parameters

If you executed the above procedure correctly, all slave units will automatically change their couple state to ON and from this moment onwards, all outputs will be synchronized. If you want to remove slave units from couple mode, all you have to do is change the Couple State on the master unit to OFF.

After you have synchronized all of your instruments, you can further adjust the phase offset between the instrument. Select the Edge Offset parameter and rotate the dial to program the required phase offset. While modifying the edge offset, bear in mind that both channels are shifted simultaneously from the adjacent channel. Also note that the offset has 4 points resolution, so expect to have increments of sample clock period x 4.

NOTE

Due to cable connections and depending on the length of the cables you are using, it is possible that you'll notice some phase delays between the master and slave units. This is perfectly normal. If you are using 1-meter cables, you should expect about 2 ns delay between master and slave. Additional delays will accumulate as you add slave units. You can minimize the delay effect by using shorter cables however, it is impossible to

eliminate this delay entirely. One way to compensate for inter-instrument delays is using different cable length at the output connectors. You can also use the edge-offset parameter to compensate for larger delays.

Saving and Recalling Setups

Needless to say how aggravating it could be to spend long time on programming front panel controls when, all of a sudden, the power fails and everything gets lost. The 8026 has battery backup that has two functions: 1) Restores front panel settings in case of power loss or after power off and 2) Retain voltage backup to memory cells where ten complete front panel setting can be preserved. The 8026 is using Ni-Cad battery circuit, which can backup the memory for almost four days.

NOTE

The 8026 is using Ni-Cad batteries to backup the memory. You must charge the batteries for 16 hours to their full capacity. Batteries are charged automatically whenever the instrument is plugged to AC power and the POWER switch is turned on.

The battery circuit backs up the waveform memory as well. However, segment tables can not be saved. Therefore, if you save front panel setups, bear in mind that only front panel controls and parameters can be stored. Waveform segments are stored for the last setting before power off only.

Refer to Figure 3-29 and Figure 3-30 and the following description to learn how to save and recall front panel setups.

1. Press the UTILITY button
2. Select and press the System option
3. Select and press the Setups option
4. Select and press the Setup Select button
5. Rotate the dial to select the required memory location
6. Press Save to store current front panel settings

Press Recall to update front panel settings with the required

---

memory location
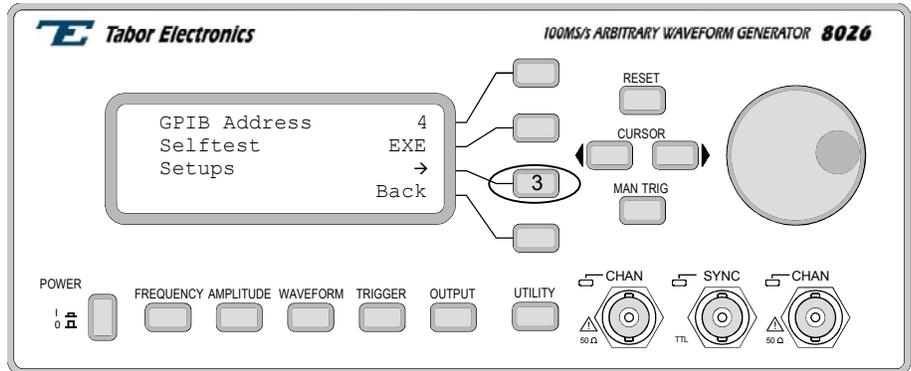
Figure 3-29. Selecting Setups from the System Menu
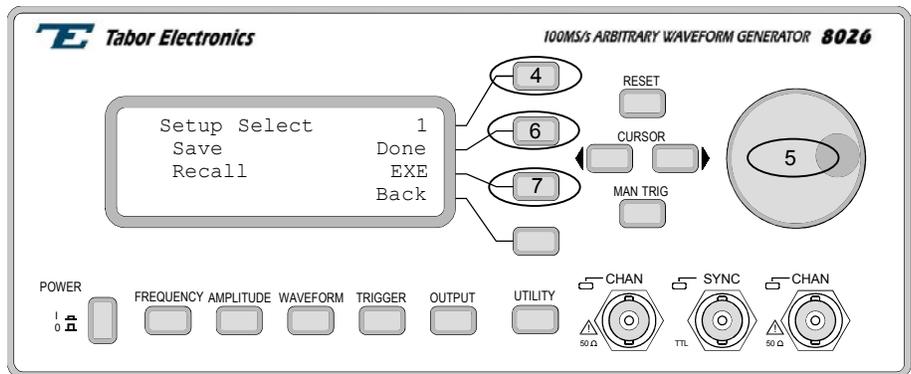
Figure 3-30. Saving and Recalling Setups

Generating Arbitrary Waveforms

In general, the Model 8026 cannot by itself create arbitrary waveforms. If you want to use arbitrary waveforms, you must first load them into the instrument. The 8026 is supplied with waveform creation and editing, called – ArbExplorer. Besides waveform generation, ArbExplorer has instrument control features, sequence table generator, FM composer and many other features that will be described separately. Figure 3-31 shows an example of a waveform that was created with the ArbExplorer. Once the waveform is created on the screen, downloading it to the 8026 is just a click of a mouse away.

Detailed information on the structure of the arbitrary waveform and the commands that are needed to download arbitrary waveforms to the 8026 is given in Chapter 4. Information in this Chapter will give

you some general idea what arbitrary waveforms are all about.
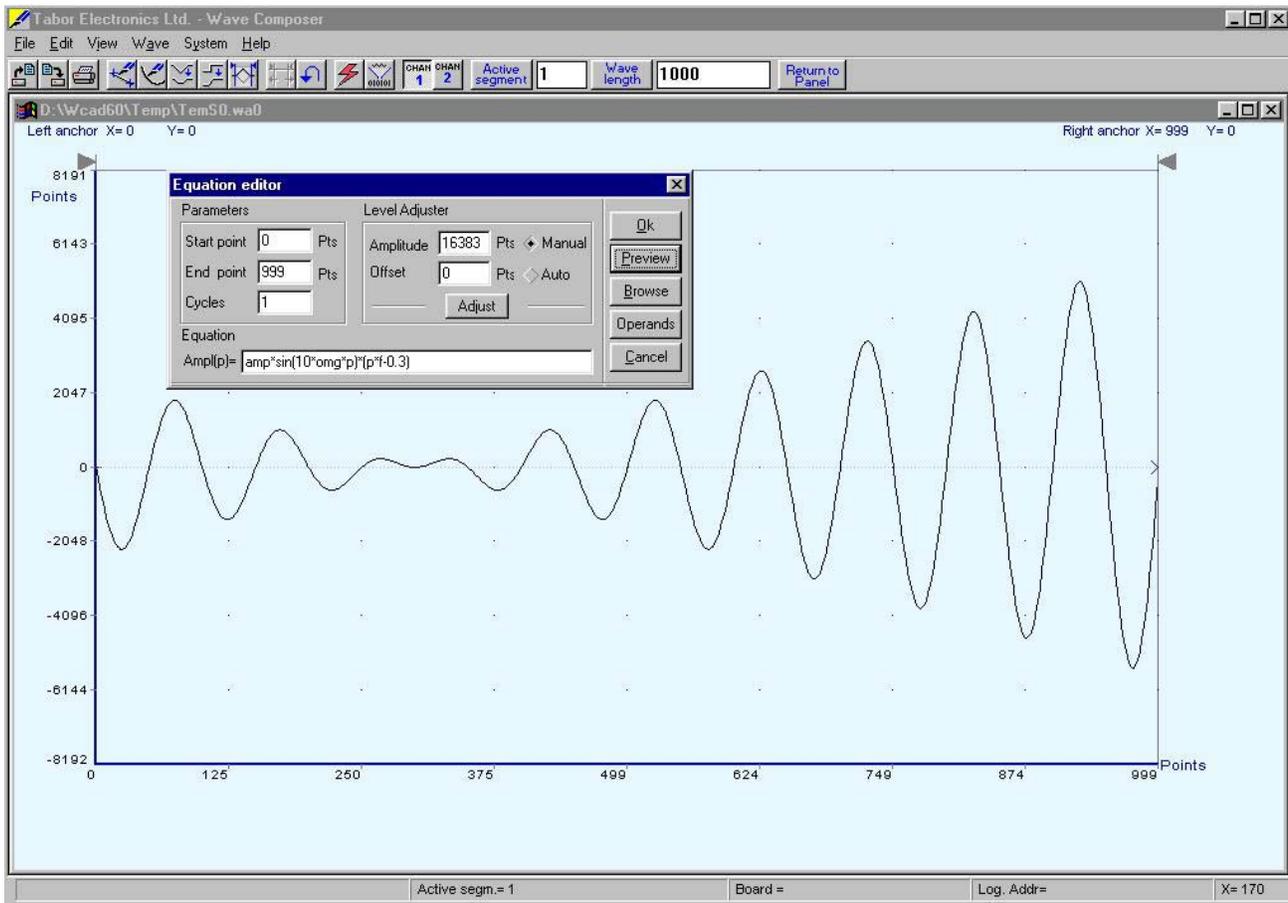
Figure 3-31. The Wave Composer Tool for Generating Arbitrary Waveforms

What Are Arbitrary Waveforms?

Arbitrary waveforms are generated from digital data points, which are stored in a working memory. The working memory is connected to a digital to analog converter (DAC) and a sample clock generator is clocking the data points, one at a time, to the output circuit. In slow motion, the output generates a waveform that resembles the look of a staircase. In reality, the DAC is generating amplitude hops that depend on bit arrangement and sample clock speed.

The working memory has two major properties: vertical resolution and memory depth.

Vertical Resolution – This term defines the precision along the vertical axis of which data points can be placed and generated by the DAC. The 8026 is using 14-bit DAC's to generate arbitrary waveforms. Converting 14 bits to precision shows that each data point can be placed along the vertical axis with a precision of

1/16,384.

Memory Depth – Defines how many data points can be stored for a single waveform cycle. The 8026 has the following waveform memory capacity:

1 Meg – standard memory configuration
4 Meg – optional memory expansion

Having large memory capacity has big advantages. Modern applications in the telecommunications industry require simulation of long waveforms without repeatable segments. The only way to create such waveforms is having sufficient memory depth. On the other hand, if you do not need to use very long waveforms but must have many other waveforms stored in your working memory, the 8026 lets you divide the memory bank to smaller segments and load different waveforms into each segment.

Using the Arbitrary Waveforms

Downloading waveforms to the 8026 and managing arbitrary memory are explained in the programming section of this manual. This section assumes that you have already downloaded waveforms and want the instrument to output these waveforms.

Refer to Figure 3-32 and use the following description to learn how to output arbitrary waveforms. Here is what you do:

1. Press the WAVEFORM button
2. Select and press the ARB option
3. Select and press the Segment CH1 button
4. Rotate the dial to select the required memory segment

If you did not make programming errors and did not make any mistake while downloading your waveform segments, then the output should generate your desired waveform. This is not all, you now have to make sure that the sample clock is programmed to the correct rate so that your output will match your frequency requirements. This is easy since we already showed you earlier how to set sample clock frequency and how to compute the output frequency from the sample clock value and the number of points you are using for your waveform.
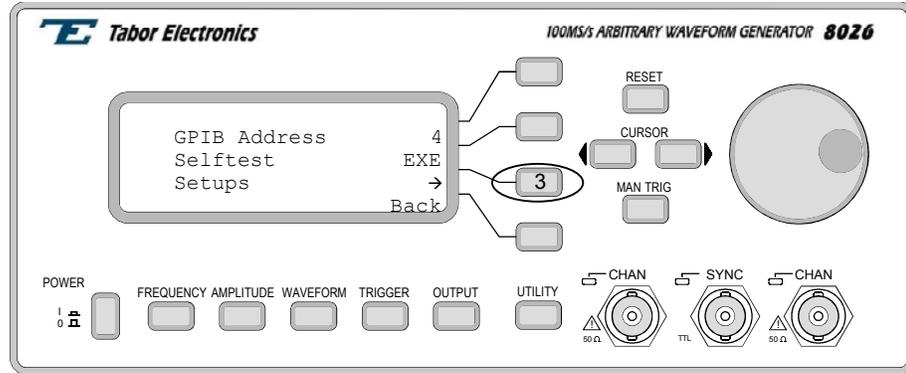
Figure 3-32. Selecting Arbitrary Waveform Segments

Generating Sequenced Waveforms

In general, the Model 8026 cannot by itself create sequenced waveforms. If you want to use sequenced waveforms, you must first load them into the instrument. The 8026 is supplied with waveform creation and editing, called – ArbExplorer. Besides waveform creation, ArbExplorer has instrument control features, sequence table generator, FM composer and many other features that will be described separately. To generate a sequence you must first download waveforms to the instrument, generate a sequence table and download the sequence table to the instrument.

Figure 3-33 shows an example of an ArbExplorer panel with an open sequence table generator. Note that different sequences can be generated for each channel.

Detailed information on the structure of the arbitrary waveform and the commands that are needed to download arbitrary waveforms to the 8026 is given in Chapter 4. You can also find in this chapter information how to create and download sequence tables using SCPI programming commands, or ArbExplorer panels. Information in this chapter will give you some general idea what sequenced waveforms are all about.
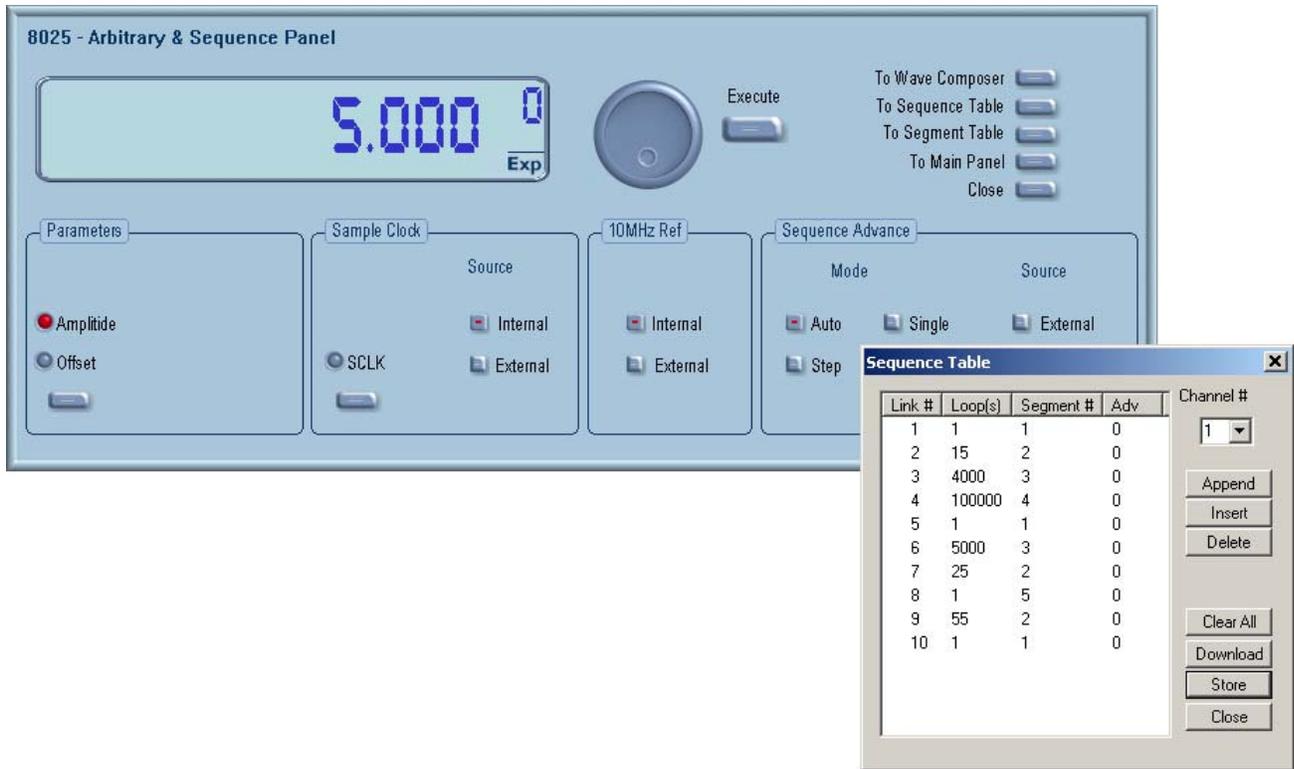
Figure 3-33. Using ArbExplorer to Generate Sequences

What Are Sequenced Waveforms?

Sequenced waveforms are constructed from two or more arbitrary waveforms, which are linked and looped in any way you can imagine, as long as you observe the limitations set forth in the specification section of this manual.

The first thing to do before you can generate sequenced waveforms is download waveforms to the 8026. You may use ArbExplorer or any other application to create waveform segments. Then, you can build your sequence table. An example of how sequenced waveforms work with three different waveforms is demonstrated in Chapter 1, figures 1.6a through 1.6d.

The sequence table specifies link #, Loop(s) and Segment #. Description of the various elements within the sequence table is given below.

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

Loop(s) – 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.

Segment # - 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 how you associate links to segments, except you can not program in the sequence table segments that were not defined earlier.

Adv – This field is a special code that is used in conjunction with the mixed advance mode. This bit flags the 8026 if the selected link is continuous or stepped. Information on the Mixed sequence advance mode is given later. "0" flags continuous, "1" flags stepped.

Assuming that you already downloaded waveforms, created and downloaded sequence table, you can proceed now with the following description how to set the 8026 to output sequenced waveforms. Refer to Figure 3-34 for the following.

1. Press the WAVEFORM button
2. Select and press the SEQ option

If you did not make any errors while downloading waveforms and creating your sequence table, then you should be able to observe your sequenced waveform at the 8026 outputs.
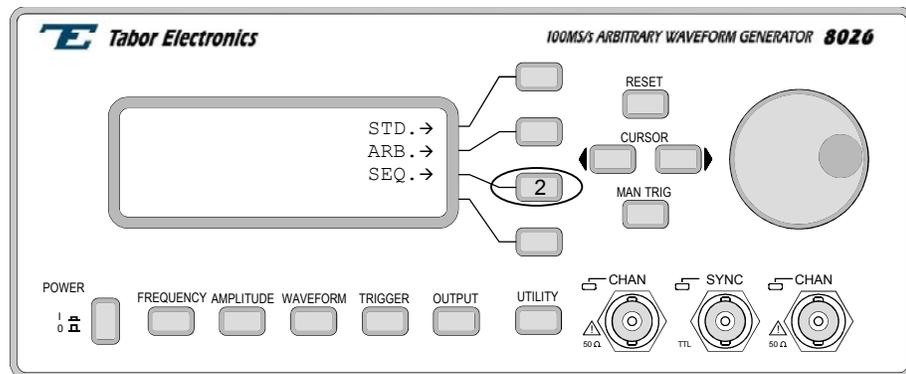
Figure 3-34. Selecting Sequenced Waveforms

Changing Sequence Advance Modes

As was explained above, the 8026 steps through an index of links; It may loop a few times on a designated link and eventually, after the last link, the process repeats itself. Stepping from link to link through the sequence table is done automatically by the instrument. However, there are application requiring control when and how the link is stepped. The 8026 has a number of sequence advance options: Auto, Stepped, Single and Mixed. These advance modes are described in the following.

Automatic – specifies continuous advance where the generator steps continuously and automatically through the links to the end of the sequence table and then repeats the sequence from the start. For example, if a sequence is made of three segments – 1, 2, and 3, and AUTO mode is used, 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. AUTO is the default sequence advance mode. Note, to use this mode, the 8026 must be in *continuous* operating mode.

Stepped – Using this advance mode, the sequence is advanced to the next link only when a valid trigger is received. The output of the 8026 generates the first segment continuously until a trigger signal advances the sequence to the next link. If repeats were selected for a segment, the loop counter is executed automatically. Note, although the trigger input controls advanced steps, to use this mode, the 8026 must be in *continuous* operating mode.

Single - Using this advance mode, the 8026 idles between steps until a valid trigger signal is sensed. The single advance mode requires that the 8026 be in trigger operating mode only. An attempt to select the Single advance mode when the instrument is in continuous operating mode can not be done. When triggered, 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 were programmed, the output will repeat this segment for n times automatically. Only after executing all of the programmed loops will the sequence step to the next assigned link. Note, to use this mode, the 8026 must be in *triggered* operating mode.

Mixed – This sequence advance mode allows combination of automatic and stepped links in one sequence table. To use this mode, in the sequence table, mark the Adv field "1" to flag stepped link or "0" for continuous link. Then, download the sequence table to the 8026. Note, to use this mode, the 8026 must be in *continuous* operating mode. Step with a "0" bit assigned to a step 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, then will wait for the next trigger to advance to the next step.

Refer to Figure 3-35 and the following description to learn how to select sequence advance modes.

1. Press the WAVEFORM button
2. Select and press the SEQ option
3. Select and press the Advance Mode option
4. Rotate the dial to select an advance mode
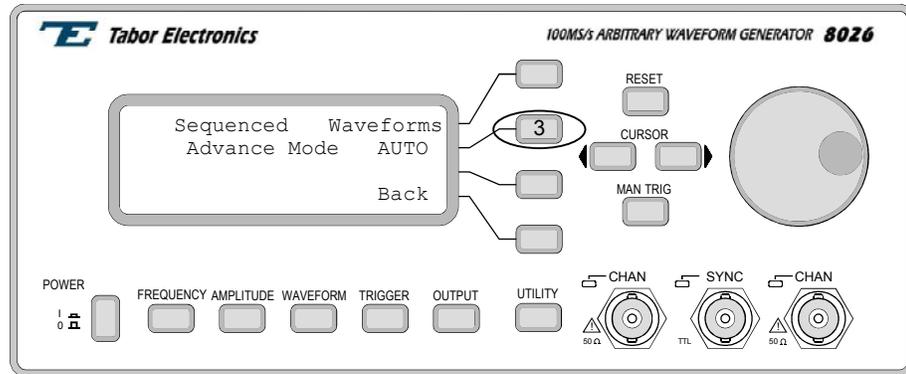
Figure 3-35. Selecting Sequence Advance Mode

NOTE

The operating mode of the instrument, as selected from the TRIGGER menu, affects the way that the 8026 lets you access the sequence advance mode parameter. If you are in continuous mode, you can select one of: AUTO, STEP, or MIXED as advance modes. If you are in triggered operating mode, you'll be able to select from AUTO, or SINGLE as advance modes.

TIP

If you are using trigger stimuli to advance links, you have a choice of either using the TRIG IN connector to apply external signals, or use the internal trigger generator. Selection of the trigger source is done from the TRIGGER menu.

Chapter 4

REMOTE 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 8026. 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. Complete listing of all commands used for programming the 8026 is given in Table 4.1.

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 Model 8026 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 consortium. The commands on the following pages are broken into three columns; the KEYWORD, the PARAMETER FORM, and any NOTES.

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 8026 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.

Table 4-1. Model 8026 SCPI Commands List Summary

Keyword	Parameter Form (Default in Bold)	Notes
:FORMat		
:BORDer	**NORMal** | SWAPped	GPIB command only
:INSTRument		
[:SELect]	**1** | 2	
:COUPle		
[:STATe]	**OFF** | ON	
:MODE	**MASTer** | SLAVe	
:PHASe		
[:OFFSet]	(**0**,0,999999)	4 point increments
:OUTPut		
[:STATe]	**OFF** | ON	
:FILTer		
[:LPASs]	**NONE** | 25M | 50M | ALL	
:SYNC		
[:STATe]	**OFF** | ON	
:SOURce	**BIT** | LCOMplete	
:POSition		
[:POINt]	(**0**;0;999999)	4 point increments
:WIDTh	(**4**;4;100000)	4 point increments
[:SOURce]		
:APPLy		
:SINusoid	FREQ,AMPL,OFFS,PHAS	
:TRIangle	FREQ,AMPL,OFFS,PHAS	
:SQUare	FREQ,AMPL,OFFS,DCYC	
:PULSe	FREQ,AMPL,OFFS,DEL,WIDT,LEAD,TRA	
:RAMP	FREQ,AMPL,OFFS,DEL,LEAD,TRA	
:SINC	FREQ,AMPL,OFFS, NCYC	
: GAUSsian	FREQ,AMPL,OFFS,EXP	
: EXPonential	FREQ,AMPL,OFFS,EXP	
: DC	DC_AMPL	
: USER	SEGM,SCLK,AMPL,OFFS	

Table 4-1. Model 8026 SCPI Commands List Summary (continued)

Keyword	Parameter Form (Default in Bold)	Notes
:FREQuency		
[:CW]	(**1e6**;100e-6;50e6) | MINimum | MAXimum	
:RASTer	(**1e7**;100e-3;100e6) | MINimum | MAXimum	
:SOURce	INTernal | EXTernal	
:DIVider	(**1**;1;65535)	Integers only
:ROSCillator		
:SOURce	**INTernal** | EXTernal	
:PHASe		
[:OFFSet]	(**0**;0;999999)	1 point increments
:VOLTage		
[:LEVel]		
[:AMPLitude]	(5.**000**;10E-3;10.00) | MINimum | MAXimum	
:OFFSet	(**0**;-4.5;+4.5)	
:FUNCtion		
:MODE	**FIXed** | USER | SEQuence	
:SHAPe	**SINusoid** | TRIangle | SQUare | PULSe | RAMP | SINC | GAUSsian | EXPonential | NOISe | DC	
:SINusoid		
:PHASe	(**0**;0;360)	
:TRIangle		
:PHASe	(**0**;0;360)	
:SQUare		
:DCYCle	(**50**;1;99)	
:PULSe		
:DELay	(**10**;0;99.9)	
:WIDth	(**10**;0;99.9)	
:TRANsition		
[:LEADing]	(**10**;0;99.9)	
:TRAiling	(**10**;0;99.9)	

Table 4-1. Model 8026 SCPI Commands List Summary (continued)

Keyword	Parameter Form (Default in Bold)	Notes
:RAMP		
:DELay	(**0**;0;99.9)	
:TRANsition		
[:LEADing]	(**60**;0;99.9)	
:TRAiling	(**30**;0;99.9)	
:GAUSsian		
:EXPonent	(**20**;10;200)	
:SINC		
:NCYCle	(**10**;4;100)	
:EXPonential		
:EXPonent	(**1**;-20.00;20.00)	
:DC		
[:AMPLitude]	(**100**;-100;100)	
:FM		
:DATA	<arbitrary_block>	
:DEViation	(**10e6**;100e-3;100e6)	
:FUNCtion		
:MODE	**FIXed** | USER	
:SHAPe	**SINusoidal** | TRIangle | SQUare | RAMP	
:FREQuency	(**1e3**;1e-3;100e3)	
:RASTer	(**1e6**;1e-3;10e6)	
[:STATe]	**OFF** | ON	
:TRIGger		
:MODE	**CONTinuous** | TRIGgered | GATed	
:SLOPe	**POSitive** | NEGative	
:FSK		
:FREQuency		
:RASTer	(**80e6**;100e-3;100e6)	
[:STATe]	**OFF** | ON	
:MODE	**HOP** | RAMP	
:RAMP		
:TIME	(1e-3;10e-6;1)	

Table 4-1. Model 8026 SCPI Commands List Summary (continued)

Keyword	Parameter Form (Default in Bold)	Notes
:SWEep		
[:FREQuency]		
:STOP	(**20e6**;100e-3;100e6)	
[:STATe]	**OFF** | ON	
:TIME	(**1e-3**;1e-3;1000)	
:DIRection	**UP** | DOWN	
:SPACing	**LINear** | LOGarithmic	
:TRIGger	**CONTinuous** | TRIGgered | GATed	
:MODE	**CONTinuous** | TRIGgered | GATed	
:SLOPe	**POSitive** | NEGative	
:MARKer	(64e6;100e-3;100e6)	
:AM		
[:STATe]	**OFF** | ON	
:TRACe		
[:DATA]	<arbitrary_block>	
DEFine	(1;1;4096),(16;16;1048326)	Even number, 1 Meg memory, divisible by 4
DEFine	(1;1;4096),(16;16;4193304)	Even number, 4 Meg memory, divisible by 4
:DELete		
[:NAME]	(1;1;4096)	
:ALL		
:SELect	(**1**;1;4096)	
:SEGMent		
[:DATA]	<binary_block>	
:DMA		
[:STATe]	OFF|ON	OFF is automatic
:TYPE	**WAVE** | FM	
:SEQuence		
[:DATA]	<binary_block>	
:ADVance	**AUTOmatic** | STEP | SINGle | MIXed	
:SOURce	**EXTernal** | INTernal	
:DEFine	(1;1;4096),(1;1;4096),(1;1;1E6),(0,0,1)	Step, segment, repeat, advance mode
:DELete		
:ALL		

Table 4-1. Model 8026 SCPI Commands List Summary (continued)

Keyword	Parameter Form (Default in Bold)	Notes
:INITiate		
[:IMMediately]		
:CONTinuous	**ON** | OFF	
:TRIGger		
:BURSt		
[:STATe]	**OFF** | ON	
:COUNt	(**1**;1;1E6)	
:SOURce		
:ADVance	**EXTernal** | INTernal	
:GATE		
[:STATe]	**OFF** | ON	
:SLOPe	**POSitive** | NEGative	
:TIMer	(**1e3**;100e-3;5e6)	In Hz units
:PHASe	(**0**;0;999999)	4 points increments
[:IMMediate]		
:ARM		
[:STATe]	**OFF** | ON	
:SLOPe		
[:STARt]	**POSitive | NEGative**	
:BREakpoint		
:POSition	(**0**;0;999999)	4 points increments
:RESet		
:SYSTem		
:ERRor?		Query only
:VERSion?		Query only, 1999.0
:SAVE	(0;0;10)	Query only
:RECall	(0;0;10)	Query only
:TEST		
[:ALL]?		Query only

Table 4-1. Model 8026 SCPI Commands List Summary (continued)

Keyword	Parameter Form (Default in Bold)	Notes
*CLS		
*ESE	(**0**;0;255)	
*OPC		
*RST		
*SRE	(**0**;0;255)	
*TRG		
*ESE?		Query only
*ESR?		Query only
*IDN?		Query only
*OPC?		Query only
*OPT?		Query only
*SRE?		Query only
*STB?		Query only
*TST?		Query only

SOURce Subsystem

This subsystem is used to control output functions, shape and parameters, frequency, amplitude and amplitude modulation, and filter type. Optional nodes were omitted from these commands. Factory defaults after *RST are shown in bold typeface. Parameter low and high limits are given where applicable.

Keyword	Parameter Form	Default,Low Limit,High Limit
[:SOURce]		
:AM (?)	{OFF|ON|0|1}	**OFF**
:APPLy (?)		
:SINusoid (?)	{<freq>,<ampl>,<offs>,<phase>}	
:TRIangle (?)	{<freq>,<ampl>,<offs>,<phase>}	
:SQUare (?)	{<freq>,<ampl>,<offs>,<duty_cycle>}	
:PULSe (?)	{<freq>,<ampl>,<offs>,<delay>,<rise>,<high>,<fall>}	
:RAMP (?)	{<freq>,<ampl>,<offs>,<delay>,<rise>,<fall>}	
:SINC (?)	{<freq>,<ampl>,<offs>,<N_cycles>}	
:EXPonential (?)	{<freq>,<ampl>,<offs>,<exp>}	
:GAUSsian (?)	{<freq>,<ampl>,<offs>,<exp>}	
:DC (?)	<%_ampl>	
:USER (?)	{<seg#>,<sclk>,<ampl>,<offs>}	
:PHASe		
:OFFSet (?)	<phase_offs>	**0**,0,999999
:FM (?)	{OFF|ON|0|1}	**OFF**
:DATA	#<header><binary_block>	
:DEViation (?)	<deviation>	**1e6**,100e-3,100e6
:FUNCtion		
:MODE (?)	{FIXed|USER}	**FIXed**
:SHAPe (?)	{SINusoid|TRIangle|SQUare|RAMP)	**SINusoid**
:FREQuency	<FM_freq>	1e3,1e-3,100e3
:RASTer (?)	<FM_sclk>	**1e6**,1e-3,10e6
:TRIGger		
:MODE (?)	{CONTinuous|TRIGgered|GATed}	**CONTinuous**
:SLOPe (?)	{POSitive|NEGative	**POSitive**
:FREQuency (?)	{<freq>|MINimum|MAXimum}	**1e6**,100e-6,50e6
:RASTer (?)	{<sclk>|MINimum|MAXimum}	**10e6**,100e-3,100e6
:SOURce (?)	{EXTernal|INTernal}	**INTernal**
:DIVider (?)	<divider>	**1**,1,65535
:FSK (?)	OFF|ON|0|1}	**OFF**
:FREQuency		
:RASTer (?)	<FSK_sclk>	**80e6**,100e-3,100e6
:MODE (?)	{HOP|RAMP}	**HOP**
:RAMP		
:TIME (?)	<time>	**1e-3**,10e-6,1
:FUNCTion		
:MODE (?)	{FIXed|USER|SEQuence}	**FIXed**
:SHAPe (?)	{SINusoid|TRIangle|SQUare|PULSe|RAMP |SINC|EXPonential|GAUSsian|NOISe|DC}	**SINusoid**
:ROCSillator		
:SOURce (?)	(INTernal|EXTernal)	**INTernal**
:SWEep (?)	OFF|ON|0|1}	**OFF**
:STOP (?)	<stop_sclk>	**20e6**,100e-3,100e6
:TIME (?)	<time>	**1e-3**,1e-3,1000
:DIRection (?)	(UP|DONW)	**UP**
:SPACing (?)	(LINear|LOGarithmic)	**LINear**
:TRIGger		
:MODE (?)	{CONTinuous|TRIGgered|GATed}	**CONTinuous**

```
   :SLOPe (?)          {POSitive|NEGative              POSitive
 :MARKer (?)           <mark_sclk>                     64e6,100e-3,100e6
:VOLTage (?)           {<ampl>|MINimum|MAXimum}        5,10e-3,10
   :OFFSet (?)         <offs>                          0,-4.5,4.5
 :SINusoid
   :PHASe (?)          <phase>                         0,0,360
 :TRIangle
   :PHASe (?)          <phase>                         0,0,360
 :SQUare
   :DCYCle (?)         <duty_cycle>                    50,1,99
 :PULSe
   :DELay (?)          <delay>                         10,0,99.9
   :WIDTh (?)          <pulse_width>                   10,0,99.9
   :TRANsition (?)     <rise>                          10,0,99.9
     :TRAiling (?)     <fall>                          10,0,99.9
 :RAMP
   :DELay (?)          <delay>                         0,0,99.9
   :TRANsition (?)     <rise>                          60,0,99.9
     :TRAiling (?)     <fall>                          30,0,99.9
 :SINC
   :NCYCle (?)         N_cycles>                       10,4,100
 :GAUSsian
   :EXPonent (?)       <exp>                           20,10,200
:EXPonential
   :EXPonent (?)       <exp>                           1,-20,+20
:DC (?)                <%_amplitude>                   100,-100,+100
```

AM{OFF|ON|0|1}
Purpose
This command turns the AM function on and off

- ON or 1 will set the AM on

- OFF or 0 will set the AM off

Parameter type
Boolean

AM?
Response and default
The 8026 will return 1 if the AM is on, or 0 if the AM is off. Default is 0.

APPLy:SINusoid{<freq>,<ampl>,<offs>,<phase>}
Purpose
This command is a high-level command that programs the 8026 to output sine waveform along with its associated parameters

- <freq> sets the output frequency in units of hertz

- <ampl> sets the output amplitude in units of volts

- <offs> sets the output offset in units of volts

- <phase> sets the output start phase in units of degrees

Parameter type
Numeric

Parameter range
<freq>   10e3 to 50e6
<ampl>   10e-3 to 10
<offs>   -4.5 to +4.5
<phase>   0 to 360

APPLy:SINusoid?
Response
The 8026 will return <freq>, <ampl>,<offs>,<phase>. Default values are 1e6,5,0,0

APPLy:TRIangle{<freq>,<ampl>,<offs>,<phase>}
Purpose
This command is a high-level command that programs the 8026 to output triangular waveform along with its associated parameters

- <freq> sets the output frequency in units of hertz

- <ampl> sets the output amplitude in units of volts

- <offs> sets the output offset in units of volts

- <phase> sets the output start phase in units of degrees

Parameter type
Numeric

Parameter range
<freq>   10e3 to 50e6
<ampl>   10e-3 to 10
<offs>   -4.5 to +4.5
<phase>   0 to 360

APPLy:TRIangle?
Response
The 8026 will return <freq>, <ampl>,<offs>,<phase>. Default values are 1e6,5,0,0

APPLy:SQUare{<freq>,<ampl>,<offs>,<duty_cycle>}

Purpose
This command is a high-level command that programs the 8026 to output square waveform along with its associated parameters

- <freq> sets the output frequency in units of hertz

- <ampl> sets the output amplitude in units of volts

- <offs> sets the output offset in units of volts

- <duty_cycle> sets the squarewave duty cycle in units of %

Parameter type
Numeric

Parameter range
<freq>   10e3 to 50e6
<ampl>   10e-3 to 10
<offs>   -4.5 to +4.5
<duty_cycle> 0 to 360

APPLy:SQUare?
Response
The 8026 will return <freq>,<ampl>,<offs>,<duty_cycle>. Default values are 1e6,5,0,50

APPLy:PULSe{<freq>,<ampl>,<offs>,<delay>,<rise>,<high>,<fall>}

Purpose
This command is a high-level command that programs the 8026 to output pulse waveform along with its associated parameters

- <freq> sets the output frequency in units of hertz

- <ampl> sets the output amplitude in units of volts

- <offs> sets the output offset in units of volts

- <delay> sets the delay to the pulse rise transition in units of %

- <rise> sets the pulse rise time in units of %

- <high> sets the pulse high time in units of %

- <fall> sets the pulse fall time in units of %

Parameter type
Numeric

Parameter range
<freq>   10e3 to 50e6
<ampl>   10e-3 to 10
<offs>   -4.5 to +4.5
<delay>   0 to 99.9
<rise>   0 to 99.9
<high>   0 to 99.9
<fall>   0 to 99.9

APPLy:PULSe?
Response
The 8026 will return <freq>, <ampl>,<offs>,<delay>,<rise>,<high>,<fall>.
Default values are 1e6,5,0,10,10,10,10

APPLy:RAMP{<freq>,<ampl>,<offs>,<delay>,<rise>,<fall>}
Purpose
This command is a high-level command that programs the 8026 to output ramp waveform along with its associated parameters

- <freq> sets the output frequency in units of hertz

- <ampl> sets the output amplitude in units of volts

- <offs> sets the output offset in units of volts

- <delay> sets the delay to the pulse rise transition in units of %

- <rise> sets the pulse rise time in units of %

- <fall> sets the pulse fall time in units of %

Parameter type
Numeric

Parameter range
<freq>   10e3 to 50e6
<ampl>   10e-3 to 10
<offs>   -4.5 to +4.5
<delay>   0 to 99.9
<rise>   0 to 99.9
<fall>   0 to 99.9

APPLy:RAMP?
Response
The 8026 will return <freq>, <ampl>,<offs>,<delay>,<high>,<fall>. Default values are 1e6,5,0,10,10,10

APPLy:SINC{<freq>,<ampl>,<offs>,<N_cycles>}
Purpose
This command is a high-level command that programs the 8026 to output sinc waveform along with its associated parameters

- <freq> sets the output frequency in units of hertz

- <ampl> sets the output amplitude in units of volts

- <offs> sets the output offset in units of volts

- <N_cycles> sets the number of zero crossing

Parameter type
Numeric

Parameter range
<freq>   10e3 to 50e6
<ampl>   10e-3 to 10
<offs>   -4.5 to +4.5
<N_cycle>   4 to 100

APPLy:SINC?
Response
The 8026 will return <freq>, <ampl>,<offs>,<N_cycle>. Default values are 1e6,5,0,10

APPLy:EXPonential{<freq>,<ampl>,<offs>,<exp>}
Purpose
This command is a high-level command that programs the 8026 to output exponential waveform along with its associated parameters

- <freq> sets the output frequency in units of hertz

- <ampl> sets the output amplitude in units of volts

- <offs> sets the output offset in units of volts

- <exp> sets the exponent value

Parameter type
Numeric

Parameter range
<freq>   10e3 to 50e6
<ampl>   10e-3 to 10
<offs>   -4.5 to +4.5
<exp>   -20 to +20

APPLy:EXPonential?
Response
The 8026 will return <freq>, <ampl>,<offs>,<exp>. Default values are 1e6,5,0,1

APPLy:GAUSsian{<freq>,<ampl>,<offs>,<exp>}
Purpose
This command is a high-level command that programs the 8026 to output gaussian waveform along with its associated parameters

- <freq> sets the output frequency in units of hertz

- <ampl> sets the output amplitude in units of volts

- <offs> sets the output offset in units of volts

- <exp> sets the exponent value

Parameter type
Numeric

Parameter range
<freq>   10e3 to 50e6
<ampl>   10e-3 to 10
<offs>   -4.5 to +4.5
<exp>   10 to 200

APPLy:GAUSsian?
Response
The 8026 will return <freq>, <ampl>,<offs>,<exp>. Default values are 1e6,5,0,20

APPLy:DC{<%_ampl>}
Purpose
This command is a high-level command that programs the 8026 to output DC level along with its associated parameter

- <%_ampl> sets the output level as percentage of the amplitude setting

Parameter type
Numeric

Parameter range
<%_ampl> -100 to +100 in units of %

APPLy:DC?
Response to query version
The 8026 will return <%_ampl>. Default values are 100

APPLy:USER{<seg# >,<sclk>,<ampl>,<offs>}
Purpose
This command is a high-level command that programs the 8026 to output user-defined waveform along with its associated parameters

- <seg# > selects the active segment number

- <sclk> sets the sample clock frequency in units of samples per second

- <ampl> sets the output amplitude in units of volts

- <offs> sets the output offset in units of volts

Parameter type
Numeric

Parameter range
<seg#>   1 to 4096
<sclk>   10e-6 to 100e6
<ampl>   10e-3 to 10
<offs>   -4.5 to +4.5

APPLy:USER?
Response
The 8026 will return <seg#>,<sclk>, <ampl>,<offs>. Default values are 1,10e6,5,0

PHASe:OFFSet<phase_offs>
Purpose
This command programs the start phase offset between channels 1 and 2 in units of waveform points. Phase offset resolution when using this command is 1 point.

- <phase_offs> sets the phase offset between the channels. Phase offset is in waveform points

Parameter type
Numeric

Parameter range
<phase_offs>   0 to 999999

PHASe:OFFSet?
Response and default
The 8026 will return the present phase offset value. Default value is 0.

FM(OFF|ON|0|1}
Purpose
This command will turn the FM function on and off.

- ON or 1 will set the FM on

- OFF or 0 will set the FM off

Parameter type
Boolean

FM?
Response and default

The 8026 will return 1 if the FM is on, or 0 if the FM is off. Default is 0.

FM:DATA#<header><binary_block>
Purpose
This command will download FM modulating waveform data in binary download. Using this command, Arbitrary FM waveform table data is loaded to the 8026 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.

```
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 5.

The generator accepts binary data as 40-bit integers, which are sent in five-byte words. Therefore, the total number of bytes is always five 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 4-1 (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.

- <binary_block>   Represents arbitrary FM waveform data.

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. The FM modulating waveform data is made of 40-bit words. However, the GPIB link has 8 data bas lines and accepts 8-bit words only. Therefore, the data has to be prepared as 40-bit words and rearranged as five 8-bit words before it can be used by the 8026 as FM modulating waveform data. Figure 4-8 shows how to prepare the 40-bit word for the FM modulating waveform.

There are a number of points you should be aware of before you start preparing the data:

1. The FM function is shared by both channels
2. The number of bytes in a complete FM modulating waveform data must divide by 5. The Model 8026 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
3. The LSBit on the last byte sets marker position. "0" = sets no marker and "1" sets marker. You can set as many markers as you want.
4. The SYNC output serves as marker output when you have the 8026 set to operate in FM mode. Normal SYNC level is TTL low. The SYNC output is set to TTL high when the marker bit is true. This way you can use the SYNC output to mark frequency occurrences during FM operation.
5. Data download is terminated with the MSBit 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:

$N = SCLK\ [Hz] \times 14.31655765$

2.   Convert the result from 1 above to 4-byte (32-bit) binary word and union with an 8-bit control word as shown in Figure 4-8 (Refer to Downloading FM Modulating Waveforms Using DMA).

3.   Use an I/O routine such as ViMoveAsync (from the VISA I/O library) to transfer binary blocks of data to the generator.

4.   Last data point is sent with MSB bit set to "1"; Data transfer is completed and CPU resumes control.

There are a number of points you should be aware of before you start preparing the data:

6.   The FM function is shared by both channels
7.   The number of bytes in a complete FM modulating waveform data must divide by 5. The Model 8026 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
8.   The LSBit on the last byte sets marker position. "0" = sets no marker and "1" sets marker. You can set as many markers as you want.
9.   The SYNC output serves as marker output when you have the 8026 set to operate in FM mode. Normal SYNC level is TTL low. The SYNC output is set to TTL high when the marker bit is true. This way you can use the SYNC output to mark frequency occurrences during FM operation.
10. Data download is terminated with the MSBit of the last byte set to 1.

FM:DEViation<deviation>
Purpose
This command will set the deviation frequency in units of samples per second.

●   <deviation> sets the phase offset between the channels. Phase offset is in waveform points

Parameter type
Numeric

Parameter range
<deviation>   1e-3 to 100e6 in units of samples per second

FM:DEViation?
Response and default
The 8026 will return the present frequency deviation value. The returned value will be in standard scientific format (for example: 100mHz would be returned as 100E-3 – positive numbers are unsigned). Default value is 1e3. Default value is 1e6.

FM:FUNCtion:MODE(FIXed|USER}
Purpose
This command will set the frequency modulating waveform mode to FIXed (standard) and USER (arbitrary) waveforms.

●   FIXed will enable a library of built in standard waveforms

●   USER will set the generator to accept arbitrary waveform from an external application

Parameter type
Discrete

FM:FUNCtion:MODE?
Response and default
The 8026 will return FIX, or USER depending on its present setting. Default is FIX.

FM:FUNCtion:SHAPe(SINusoid|TRIangle|SQUare|RAMP}

Purpose

This command will select one of the waveform shapes as the active modulating waveform.

- SINusoid will select sine shape as modulating waveform

- TRIangle will select triangular shape as modulating waveform

- SQUare will select square shape as modulating waveform

- RAMP will select ramp shape as modulating waveform

Parameter type

Discrete

FM:FUNCtion:SHAPe?

Response and default

The 8026 will return SIN, TRI, SQU, or RAMP depending on its present setting. Default is SIN.

FM:FREQuency<FM_freq>

Purpose

This command will set the modulating wave frequency. This parameter affects the 8026 when set to operate in FM:FUNC:MODE FIX

- <FM_freq>   will set the frequency of the modulating waveform

Parameter type

Numeric

Parameter range

<FM_freq>   1e-3 to 100e6 in units of hertz

FM:FREQuency?

Response and default

The 8026 will return 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). Default value is 1e3.

FM:FREQuency:RASTer(?)<FM_sclk>

Purpose

This command will set the modulating wave sample clock frequency. This parameter affects the 8026 when set to operate in FM:FUNC:MODE USER

- <FM_sclk>   will set the sample clock frequency of the modulating waveform

Parameter type

Numeric

Parameter range

<FM_sclk>   1e-3 to 10e6 in units of samples per second

FM:FREQuency:RASTer?
Response and default
The 8026 will return the present modulating waveform sample clock frequency value. The returned value will be in standard scientific format (for example: 1KHz would be returned as 1E3 – positive numbers are unsigned). Default value is 1e6.

FM:TRIGger:MODE(CONTinuous|TRIGered|GATEd}
Purpose
This command will select one of the FM modes.

● CONTinuous will select continuous modulation

● TRIGered will select triggered modulation

● GATEd will select gated modulation

Parameter type
Discrete

FM:TRIGger:MODE?
Response and default
The 8026 will return CONT, TRIG, or GATE depending on its present setting. Default is CONT.

FM:TRIGger:SLOPe(POSitive|NEGative}
Purpose
This command will select slope sensitivity for the FM trigger input.

● POSitive will select rising edge

● NEGative will select falling edge

Parameter type
Discrete

FM:TRIGger:SLOPe?
Response and default
The 8026 will return POS, or NEG depending on its present setting. Default is POS.

FREQuency{<freq>|MINimum|MAXimum}
Purpose
This command modifies the frequency of the standard waveform in units of hertz (Hz). It has no affect on arbitrary waveforms.

● <freq> sets the waveform frequency

● <MINimum> sets the waveform frequency to its lowest value

● <MAXimum> sets the waveform frequency to its highest value

Parameter type
Numeric

Parameter range
<freq>   100e-6 to 50e6

FREQuency?
Response and default
The 8026 will return the present frequency value. The returned value will be in standard scientific format (for example: 100mHz would be returned as 100E-3 – positive numbers are unsigned). Default value is 1e6.

FREQuency:RASTer{<sclk>|MINimum|MAXimum}
Purpose
This command modifies the sample clock frequency of the arbitrary waveform in units of samples per second (S/s). It has no affect on standard waveforms.

- <sclk> sets the sample clock frequency
- <MINimum> sets the sample clock frequency to its lowest value
- <MAXimum> sets the sample clock frequency to its highest value

Parameter type
Numeric

Parameter range
<sclk>   100e-3 to 100e6

FREQuency:RASTer?
Response and default
The 8026 will return the present sample clock frequency value. The returned value will be in standard scientific format (for example: 100MHz would be returned as 100E6 – positive numbers are unsigned). Default value is 10e6.

FREQuency:RASTer:SOURce{EXTernal|INTernal}
Purpose
This command selects the source of the sample clock generator. This command affects both the standard and the arbitrary waveforms.

- <EXTernal> selects an external source
- <INTernal> selects the internal source

Parameter type
Discrete

FREQuency:RASTer:SOURce?
Response and default
The 8026 will return EXT if an external source is selected, or INT if the internal source is selected. Default value is INT.

FREQuency:RASTer:DIVider<divider>
Purpose
The sample clock frequency for channel 2 can be divided by n. In this case, the sample clock for channel 1 will be X and the sample clock for channel 2 will be X/n. This command sets the dividing ratio for channel 2.

- <divider> sets the dividing ratio for channel 2

Parameter type
Numeric

Parameter range
<divider>   1 to 65535

FREQuency:RASTer:DIVider?
Response and default
The 8026 will return the present channel 2 sample clock frequency divider value. Default value is 1.

FSK:FREQuency:RASTer<FM_sclk>
Purpose
This command will set the shift sample clock frequency. When set to operate in FSK mode, the 8026 will hop from carrier sample clock frequency to shifted sample clock frequency.

•   <FSK_sclk>   will set the shifted sample clock frequency for the FSK mode

Parameter type
Numeric

Parameter range
<FSK_sclk>   100e-3 to 100e6 in units of samples per second

FSK:FREQuency:RASTer?
Response and default
The 8026 will return the present shifted sample clock frequency value. The returned value will be in standard scientific format (for example: 1KHz would be returned as 1E3 – positive numbers are unsigned). Default value is 80e3.

FSK:MODE(HOP|RAMPed}

Purpose
This command will select one of the FSK modes.

•   HOP will select non-delayed hops from carrier sclk to shifted sclk

•   RAMPed will select ramped shifts from carrier sclk to shifted sclk

Parameter type
Discrete

FSK:MODE?
Response and default
The 8026 will return HOP, or RAMP depending on its present setting. Default is HOP.

FSK:RAMP:TIME<time>
Purpose
This command will set the time it takes for the sample clock frequency to ramp up and down from carrier to shifted frequencies and back. This parameter will effect the 8026 only when the instrument is set to operate in ramped FSK mode.

- <time>   will set the ramp up and down time

Parameter type
Numeric

Parameter range
<time>   10e-6 to 1 in units of seconds

FSK:RAMP:TIME?
Response and default
The 8026 will return the present ramp time value. The returned value will be in standard scientific format (for example: 10ms would be returned as 10E-3 – positive numbers are unsigned). Default value is 1e-3.

FUNCTion:MODE{FIXed|USER|SEQuence}
Purpose
This command defines the type of waveform that will be available at the output connector.

- <FIXed> selects standard waveforms

- <USER> selects arbitrary waveforms

- <SEQuence> selects sequenced waveforms

Parameter type
Discrete

FUNCTion:MODE?
Response and default
The 8026 will return FIX, USER, or SEQ depending on the present 8026 setting. Default value is FIX.

FUNCtion:SHAPe{SINusoid|TRIangle|SQUare|PULSe|RAMP|SINC|EXPonential| GAUSsian|NOISe|DC}
Purpose
This command defines the shape of waveform that will be available at the output connector. This command has no affect if present 8026 setting is USER or SEQ.

- <SINusoid> selects sine waveform

- <TRIangle> selects triangular waveform

- <SQUare> selects square waveform

- <PULSe> selects pulse waveform

- <RAMP> selects ramp waveform

- <SINC> selects sinc waveform

- <EXPonential> selects exponential waveform

- <GAUSsian selects gaussian waveform

- <NOISe> selects random noise waveform

- <DC> selects dc waveform

Parameter type
Discrete

FUNCtion:SHAPe?

Response and default
The 8026 will return SIN, TRI, SQU, PULS, RAMP, SINC, EXP, GAUS, NOIS, or DC depending on the present 8026 setting. Default value is SIN.

ROSCillator:SOURce{INTernal|EXTernal}
Purpose
This command will select the reference source for the sample clock generator.

- <INTernal> selects an internal source. The internal source is 1 ppm TCXO as standard.

- <EXTernal> activates the external reference input. An external reference must be connected to the 8026 for it to continue normal operation

Parameter type
Discrete

ROSCillator:SOURce?
Response and default
The 8026 will return INT, or EXT depending on the present 8026 setting. Default value is INT.

SWEep:STOP<stop_sclk>
Purpose
This command will set the stop sample clock frequency. When set to operate in sweep mode, the 8026 will sweep from carrier sample clock frequency to sweep stop sample clock frequency.

- <stop_sclk>   will set the sweep stop sample clock frequency for the sweep mode

Parameter type
Numeric

Parameter range
<stop_sclk>   100e-3 to 100e6 in units of samples per second

SWEep:STOP?
Response and default
The 8026 will return the present sweep stop sample clock frequency value. The returned value will be in standard scientific format (for example: 1KHz would be returned as 1E3 – positive numbers are unsigned). Default value is 20e6.

SWEep:TIME<time>
Purpose
This command will set the time it takes for the sample clock frequency to sweep from carrier to stop frequencies. This parameter will effect the 8026 only when the instrument is set to operate in sweep mode.

- <time>   will set the sweep time

Parameter type
Numeric

Parameter range
<time>   1e-3 to 1000 in units of seconds

SWEep:TIME?
Response and default
The 8026 will return the present sweep time value. The returned value will be in standard scientific format (for example: 10ms would be returned as 10E-3 – positive numbers are unsigned). Default value is 1e-3.

SWEep:DIRection{UP|DOWN}
Purpose
This command will select the sweep direction up and down.

- <UP> selects an up direction. The sample clock will sweep from start frequency set by the sclk parameter to stop frequency set by the stop sclk parameter

- <DOWN> selects the down direction. The sample clock will sweep from stop frequency set by the stop sclk parameter to start frequency set by the sclk parameter

Parameter type
Discrete

SWEep:DIRection?
Response and default
The 8026 will return UP, or DOWN depending on the present 8026 setting. Default value is UP.

SWEep:SPACing{LINear|LOGarithmic}
Purpose
This command will select the sweep spacing from linear spacing and logarithmic spacing.

- <LINear> will select linear steps

- <LOGarithmic> will select logarithmic steps

Parameter type
Discrete

SWEep:SPACing?
Response and default
The 8026 will return LIN, or LOG depending on the present 8026 setting. Default value is LIN.

SWEep:TRIGger:MODE(CONTinuous|TRIGered|GATEd}
Purpose
This command will select one of the sweep modes.

- CONTinuous will select continuous sweep modulation

- TRIGered will select triggered sweep modulation

- GATEd will select gated sweep modulation

Parameter type
Discrete

SWEep:TRIGger:MODE?
Response and default

The 8026 will return CONT, TRIG, or GATE depending on its present setting. Default is CONT.

SWEep:TRIGger:SLOPe(POSitive|NEGative}

Purpose

This command will select slope sensitivity for the sweep trigger input.

- POSitive will select rising edge

- NEGative will select falling edge

Parameter type

Discrete

SWEep:TRIGger:SLOPe?

Response and default

The 8026 will return POS, or NEG depending on its present setting. Default is POS.

SWEep:MARKer<mark_sclk>

Purpose

This command will set the marker sample clock frequency for the sweep mode.

- <mark_sclk>   will set the marker sample clock frequency

Parameter type

Numeric

Parameter range

<mark_sclk>   100e-3 to 100e6 in units of samples per second

SWEep:MARKer?

Response and default

The 8026 will return the present sweep sample clock frequency value. The returned value will be in standard scientific format (for example: 1KHz would be returned as 1E3 – positive numbers are unsigned). Default value is 64e6.

VOLTage{<ampl>|MINimum|MAXimum}

Purpose

This command programs the peak to peak amplitude of the output waveform. The amplitude is calibrated when the source impedance is 50Ω.

- <ampl> sets the amplitude

- <MINimum> sets the amplitude to its lowest value

- <MAXimum> sets the amplitude to its highest value

Parameter type

Numeric

Parameter range

<ampl>   10e-3 to 10 in units of volts
<MINimum>   10e-3 in units of volts
<MAXimum>   10 in units of volts

VOLTage?
Response to query version
The 8026 will return 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). Default value is 5.

VOLTage:OFFSet<offs>
Purpose
This command programs the amplitude offset of the output waveform. The offset is calibrated when the source impedance is 50Ω.

- <offs> sets the amplitude offset

Parameter type
Numeric

Parameter range
<offs>   -4.5 to +4.5 in units of volts

VOLTage:OFFSet?
Response and default
The 8026 will return the present offset value. The returned value will be in standard scientific format (for example: 100mV would be returned as 100E-3 – positive numbers are unsigned). Default value is 0.

SINusoid:PHASe<phase>
Purpose
This command programs start phase of the standard sine waveform. This command has no affect on arbitrary waveforms.

- <phase> sets the start phase

Parameter type
Numeric

Parameter range
<phase>   0 to 360 in units of degrees

SINusoid:PHASe?
Response and default
The 8026 will return the present start phase value. Default is 0.

TRIangle:PHASe<phase>
Purpose
This command programs start phase of the standard triangular waveform. This command has no affect on arbitrary waveforms.

- <phase> sets the start phase

Parameter type
Numeric

Parameter range
<phase>   0 to 360 in units of degrees

TRIangle:PHASe?
Response and default
The 8026 will return the present start phase value. Default value is 0.

SQUare:DCYCle<duty_cycle>
Purpose
This command programs duty cycle of the standard square waveform. This command has no affect on arbitrary waveforms.

● <duty_cycle> sets the duty cycle value

Parameter type
Numeric

Parameter range
<duty_cycle>   0 to 99 in units of percent

SQUare:DCYCle?
Response and default
The 8026 will return the present duty cycle value. Default value is 50.

PULSe:DELay<delay>
Purpose
This command programs delay of the standard pulse waveform. This command has no affect on arbitrary waveforms.

● <delay> sets the delay parameter

Parameter type
Numeric

Parameter range
<delay>   0 to 99.9 in units of percent

PULSe:DELay?
Response and default
The 8026 will return the present delay value. Default value is 10.

PULSe:WIDTh<pulse_width>
Purpose
This command programs pulse high portion of the standard pulse waveform. This command has no affect on arbitrary waveforms.

● <pulse_width> sets the high time parameter

Parameter type
Numeric

Parameter range

<pulse_width>   0 to 99.9 in units of percent

PULSe:WIDTh?
Response and default
The 8026 will return the present width value. Default value is 10.

PULSe:TRANsition<rise>
Purpose
This command programs pulse transition from low to high of the standard pulse waveform. This command has no affect on arbitrary waveforms.

● <rise> sets the rise time parameter

Parameter type
Numeric

Parameter range
<rise>   0 to 99.9 in units of percent

PULSe:TRANsition?
Response and default
The 8026 will return the present rise time value. Default value is 10.

PULSe:TRANsition:TRAiling<fall>
Purpose
This command programs pulse transition from high to low of the standard pulse waveform. This command has no affect on arbitrary waveforms.

● <fall> sets the fall time parameter

Parameter type
Numeric

Parameter range
<fall>   0 to 99.9 in units of percent

PULSe:TRANsition:TRAiling?
Response and default
The 8026 will return the present fall time value. Default value is 10.

RAMP:DELay<delay>
Purpose
This command programs delay of the standard ramp waveform. This command has no affect on arbitrary waveforms.

● <delay> sets the delay parameter

Parameter type
Numeric
Parameter range
<delay>   0 to 99.9 in units of percent

RAMP:DELay?
Response and default
The 8026 will return the present delay value. Default value is 0.

RAMP:TRANsition<rise>
Purpose
This command programs ramp transition from low to high of the standard ramp waveform. This command has no affect on arbitrary waveforms.

● <rise> sets the rise time parameter

Parameter type
Numeric

Parameter range
<rise>   0 to 99.9 in units of percent

RAMP:TRANsition?
Response and default
The 8026 will return the present rise time value. Default value is 60.

RAMP:TRANsition:TRAiling<fall>
Purpose
This command programs ramp transition from high to low of the standard ramp waveform. This command has no affect on arbitrary waveforms.

● <fall> sets the fall time parameter

Parameter type
Numeric

Parameter range
<fall>   0 to 99.9 in units of percent

RAMP:TRANsition:TRAiling?
Response and default
The 8026 will return the present fall time value. Default value is 30.

SINC:NCYCleN_cycles>
Purpose
This command programs the number of "0-crossings" of the standard SINC pulse waveform. This command has no affect on arbitrary waveforms.

● <N_cycle> sets the number of zero-crossings parameter

Parameter type
Numeric (integer only)

Parameter range
<N_cycle>   4 to 100 zero crossings

SINC:NCYCle?

Response and default
The 8026 will return the present number of zero-crossing value. Default value is 10.

GAUSsian:EXPonent<exp>
Purpose
This command programs the exponent for the standard gaussian pulse waveform. This command has no affect on arbitrary waveforms.

● <exp > sets the exponent parameter

Parameter type
Numeric

Parameter range
<exp>   10 to 200

GAUSsian:EXPonent?
Response and default
The 8026 will return the present exponent value. Default value is 20.

EXPonential:EXPonent<exp>
Purpose
This command programs the exponent for the standard exponential waveform. This command has no affect on arbitrary waveforms.

● <exp > sets the exponent parameter

Parameter type
Numeric

Parameter range
<exp>   -100 to 100

EXPonential:EXPonent?
Response and default
The 8026 will return the present exponent value. Default value is –20.

DC<%_amplitude>
Purpose
This command programs the amplitude of the standard DC waveform. This command has no affect on arbitrary waveforms.

● <%_amplitude > sets the dc level as percentage of the programmed amplitude setting

Parameter type
Numeric

Parameter range
<%_amplitude> -100 to 100 in units of percent

DC?
Response and default
The 8026 will return the present dc value. Default value is 100.

OUTPut Subsystem

This subsystem controls the characteristics of the output. It controls filter type and filter cutoff frequency. It is also used to control the sync output, source and position. Optional nodes were omitted from these commands. Factory defaults after *RST are shown in bold typeface. Parameter low and high limits are given where applicable.

Keyword	Parameter Form	**Default,**Low Limit,High Limit
:OUTPut (?)	{OFF|ON|0|1}	**OFF**
:FILTer (?)	{NONE|25M|50M|ALL}	**NONE**
:SYNC (?)	{OFF|ON|0|1}	**OFF**
:SOURce (?)	{BIT|LCOMplete}	**BIT**
:POSition (?)	\<position\>	**0,**0,999999
:WIDTh (?)	\<width\>	**4,**4,100000

OUTPut{OFF|ON|0|1}
Purpose
This command will turn the 8026 output on and off.

- ON or 1 will set the output on

- OFF or 0 will set the output off

Parameter type
Discrete

OUTPut?
Response and default
The 8026 will return 1 if the output is on, or 0 if the output is off. Default value is 0.

OUTPut:FILTer{NONE|25M|50M|ALL}
Purpose
This command will select which filter is connected to the 8026 output. Observe the following restrictions when you try to use this command:

1) 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.
2) Filters are placed before the output amplifier. Therefore, do not expect the filters to remove in-band amplifier harmonics and spurious.

- NONE will disconnect all filters from the output path

- 25M will connect 25 MHz low pass elliptic filter

- 50M will connect 50 MHz low pass elliptic filter

- ALL will connect both the 25 MHz and the 50 MHz filters in series

Parameter type
Discrete

OUTPut:FILTer?
Response and default
The 8026 will return NONE, 25M, 50M, or ALL depending on the type of filter presently connected to the output. Default value is NONE.

OUTPut:SYNC{OFF|ON|0|1}
Purpose
This command will turn the 8026 SYNC output on and off.

- ON or 1 will set the SYNC output on
- OFF or 0 will set the SYNC output off

Parameter type
Discrete

OUTPut:SYNC?
Response and default
The 8026 will return 1 if the SYNC output is on, or 0 if the SYNC output is off. Default value is OFF.

OUTPut:SYNC:SOURce{BIT|LCOMplete}
Purpose
This command will select the 8026 SYNC option.

- BIT will set the SYNC to output a narrow pulse
- LCOMplete will set the SYNC to transition high at the beginning of the sequence and low a the end of the sequence

Parameter type
Discrete

OUTPut:SYNC:SOURce?
Response and default
The 8026 will return BIT, or LCOM depending on the present setting of the SYNC source. Default value is BIT.

OUTPut:SYNC:POSition<position>
Purpose
This command will program the 8026 SYNC position. This command is active in arbitrary (USER) mode only.

- <position> will set the SYNC position in waveform points

Parameter type
Numeric (integer only)

Parameter range

<position>   0 to 999999 in units of waveform points. The sync position can be programmed in increments of 4 points minimum.

OUTPut:SYNC:POSition?
Response and default
The 8026 will return the present duty cycle value. Default value is 50.

Response to query version
The 8026 will return the present SYNC position value. Default value is 0.

OUTPut:SYNC:WIDTh<width>
Purpose
This command will program the width of the SYNC pulse. The sync width can be varied through a limited range to allow widening of the sync pulse.

● <width> will set the SYNC width

Parameter type
Numeric (integer only)

Parameter range
<width>   4 to 100000 in units of waveform points.

OUTPut:SYNC:WIDTh?
Response and default
The 8026 will return the present SYNC width value. Default value is 4.

FORMat Subsystem

The FORMat subsystem sets a data format for transferring array information. This data format is used for commands that are specifically designated to be affected by the FORMat subsystem. Factory defaults after *RST are shown in bold typeface. Parameter low and high limits are given where applicable.

Keyword	Parameter Form	Default,Low Limit,High Limit
:FORMat		
:BORDer (?)	{NORMal|SWAPped}	**NORMal**

FORMat:BORDer{NORMal|SWAPped}
Purpose
This command will set the byte order for the trace command.

- NORMal will set high-low byte order
- SWAPped will set low-high byte order

Parameter type
Discrete

FORMat:BORDer?
Response to query version
The 8026 will return NORM, or SWAP depending on the present byte order setting. Default value is NORM.

INSTrument Subsystem

A dual channel arbitrary waveform generator is considered as two logical instruments. The INSTrument subsystem provides a mechanism to identify and select instrument channels. Factory defaults after *RST are shown in bold typeface. Parameter low and high limits are given where applicable.

Keyword	**Parameter Form**	**Default,**Low Limit,High Limit
:INSTrument (?)	{1|2}	**1**
:COUPle (?)	{OFF|ON|0|1}	**OFF**
:MODE (?)	(MASTer|SLAVe)	**MASTer**
:PHASe (?)	\<phase\>	**0**,0,999999

INSTrument{1|2}
Purpose
This command will set the active channel for future programming sequences. Subsequent commands affect the selected channel only.

- 1 will set channel 1 active

- 2 will set channel 2 active

Parameter type
Discrete

INSTrument?
Response and default
The 8026 will return 1, or 2 depending on the present active channel setting. Default value is 1.

INSTrument:COUPle{OFF|ON|0|1}
Purpose
This command will enable daisy-chained instruments to operate in multi-instrument synchronization mode. Observe the following pre-requisite conditions:

1) 9-pin master/slave cable must be connected between the instruments. The Master end of the cable connected to the master unit and the slave end of the cable connected to the slave unit.
2) The SCLK OUT connector from the master unit is connected to the SCLK IN on the slave unit
3) The master unit is selected as master and the slave unit is selected as slave.

- ON or 1 will set the couple mode on

- OFF or 0 will set the couple mode off

Parameter type
Discrete

INSTrument:COUPle?
Response and default
The 8026 will return 0, or 1 depending on the present instrument couple setting. Default value is 0.

INSTrument:COUPle:MODE{MASTer|SLAVe}
Purpose
This command will select master and slave instruments in multi-instrument synchronization mode. Note that only one instrument in a chain may be set as master. All other instruments must be set as slaves.

* MASTer will set the master instrument
* SLAVe will set the slave instrument

Parameter type
Discrete

INSTrument:COUPle:MODE?
Response and default
The 8026 will return MAST, or SLAV depending on the present instrument couple setting. Default value is MAST.

INSTrument:COUPle:PHASe<phase>
Purpose
This command will program the phase offset between master and slave units. This command has no effect on the instrument in modes other than instrument couple.

* <phase> will set the leading edge offset between master and slave units

Parameter type
Numeric (integer only)

Parameter range
   <phase>   0 to 999999 in units of waveform points. The phase offset can be programmed in
                   increments of 4 points.

INSTrument:COUPle:PHASe?
Response to query version
The 8026 will return the present phase offset value. Default value is 0.

TRIGger Subsystem

The TRIGger subsystem is used to synchronize device actions with external events. These commands control the trigger modes of the Model 8026. The generator can be placed in Triggered, Gated or Burst mode. Trigger source is selectable from an external source, an internal trigger generator or a software trigger. Optional nodes were omitted from these commands. Factory defaults after *RST are shown in bold typeface. Parameter low and high limits are given where applicable.

Keyword	Parameter Form	Default,Low Limit,High Limit
:ARM (?)	{OFF|ON|0|1}	**ON**
:SLOPe	(POSitive|NEGative)	**POSitive**
:BREakpoint		
:POSition (?)	\<position>	**0,**0,999999
:INITiate		
:CONTinuous (?)	{OFF|ON|0|1}	**ON**
:TRIGger		
:BURSt (?)	{OFF|ON|0|1}	**OFF**
:COUNt ?)	\<count>	**1,**1,1e6
:GATE (?)	{OFF|ON|0|1}	**OFF**
:PHASe (?)	\<phase>	**0,**0,999999
:SLOPe (?)	{POSitive|NEGative}	**POSitive**
:SOURce		
:ADVance (?)	{EXTernal|INTernal}	**EXTernal**
:TIMer (?)	\<interval>	**1e3,**100e-3,5e6
:IMMediate		
*TRG		

ARM{OFF|ON|0|1}
Purpose
This command will arm the 8026 to stop and start generating output waveforms. The instrument will stop at a breakpoint set with one of the arm command.

- ON or 1 will set the arm mode on
- OFF or 0 will set the arm mode off

Parameter type
Discrete

ARM?
Response and default
The 8026 will return 1 if the arm mode is on, or 0 if the arm mode is off. Default value is 0.

ARM:SLOPe{POSitive|NEGative}

Purpose

This command will set the edge sensitivity of which the 8026 will stop in arm mode. The stop and start commands are applied at the rear-panel TRIG IN connector.

* POSitive will set the positive edge

* NEGative will set the negative edge

Parameter type

Discrete

ARM:SLOPe?

Response and default

The 8026 will return POS, or NEG depending on the present setting. Default value is POS.

ARM:BREakpoint:POSition<position>

Purpose

This command will program the breakpoint position for the arm function.

* <position> will set the breakpoint position

Parameter type

Numeric (integer only)

Parameter range

<position>   0 to 999999 in units of waveform points. The breakpoint position can be programmed in increments of 4 points minimum.

ARM:BREakpoint:POSition?

Response and default

The 8026 will return the present breakpoint value. Default value is 0.

INITiate:CONTinuous{OFF|ON|0|1}

Purpose

This command will set the output in continuous operation and interrupted operation. The trigger subsystem commands will affect the 8026 only after it will be set to interrupted operation.

* ON or 1 will set the interrupted mode

* OFF or 0 will set the continuous mode

Parameter type

Discrete

INITiate:CONTinuous?

Response to query version

The 8026 will return 1 if the output is interrupted, or 0 if the output is continuous. Default value is 0.

TRIGger:BURSt{OFF|ON|0|1}
Purpose
This command will set the burst mode. This command will affect the 8026 only after it will be set to interrupted operation with the command INIT:CONT OFF.

* ON or 1 will set the burst mode on
* OFF or 0 will set the burst mode off

Parameter type
Discrete

TRIGger:BURSt?
Response and default
The 8026 will return 1 if the burst is on, or 0 if the burst is off. Default value is 0.

TRIGger:COUNt<count>
Purpose
This command will set the burst counter.

* <count> will set the count number

Parameter type
Numeric (integer only)

Parameter range
  <count>   1 to 1 Meg

TRIGger:COUNt?
Response and default
The 8026 will return the present count value. Default value is 1.

TRIGger:GATE{OFF|ON|0|1}
Purpose
This command will set the gate mode. This command will affect the 8026 only after it will be set to interrupted operation with the command INIT:CONT OFF.

* ON or 1 will set the gate mode on
* OFF or 0 will set the gate mode off

Parameter type
Discrete

TRIGger:GATE?
Response and default
The 8026 will return 1 if the gate is on, or 0 if the gate is off. Default value is 0.

TRIGger:SLOPe{POSitive|NEGative}
Purpose
This command will set the edge sensitivity for the trigger input.

* <POSitive> will set the positive edge
* <NEGative> will set the negative edge

Parameter type
Discrete

TRIGger:SLOPe?
Response and default
The 8026 will return POS, or NEG depending on the present trigger setting. Default value is POS.

TRIGger:SOURce:ADVance{EXTernal|INTernal}
Purpose
This command will set the advance source for the trigger mode.

- <EXTernal> will select the external input
- <INTernal > will select the internal trigger generator

Parameter type
Discrete

TRIGger:SOURce:ADVance?
Response and default
The 8026 will return EXT, or INT depending on the present trigger source setting. Default value is EXT.

TRIGger:TIMer<interval>
Purpose
This command will set the period for the internal trigger generator.

- <interval> will set the timer of the internal trigger generator

Parameter type
Numeric

Parameter range
<interval>   100e-3 to 5e6 in units of seconds

TRIGger:TIMer?
Response and default
The 8026 will return the present internal trigger timer value. The returned value will be in standard scientific format (for example: 10ms would be returned as 10E-3 – positive numbers are unsigned). Default value is 1e3.

TRIGger:IMMediate
Purpose
This command will stimulate the 8026 output once. This command will affect the 8026 only after it will be set to interrupted operation.

*TRG
Purpose
This common command has the same affect as the TRIG:IMM command.

TRACe Subsystem

The TRACe subsystem contains commands, which allow definition of segments and their corresponding lengths, addition and deletion of segments, and the loading of waveform data. Sequence commands control segments link and loops. DMA command places 8026 in a special data transfer mode where the generator's message-based interface is bypassed and data is loaded directly from the data bus. Optional nodes were omitted from these commands. Defaults are shown in bold.

Keyword	Parameter Form	Default,Low Limit,High Limit
:TRACe	#<header><binary_block>	
:DEFine	<segment_number>,<length>	
:DELete	<segment_number>	
:DELete:ALL		
:SELect (?)	<segment_number>	
:SEQuence	#<header><binary_block>	
:ADVance (?)	{AUTOmatic|STEP|SINGle|MIXed}	**AUTOmatic**
:SOURce (?)	{EXTernal|INTernal}	**EXTernal**
:DEFine	<link>,<seg_#>,<loop>,<mode>	
:DELete:ALL		
:DMA	(OFF|ON)	**OFF**
:TYPE (?)	(WAVE|FM)	**WAVE**

Generating Arbitrary Waveforms

Arbitrary waveforms are generated from digital data points, which are stored in memory. Each data point has a vertical resolution of 14 bits (16384 points), i.e., each sample is placed on the vertical axis with a precision of 1/16384. The Model 8026 has the following waveform memory capacity:

     1 Meg – standard memory configuration
     4 Meg – optional memory expansion

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. The 8026 provides programmable sample clock rates from 100 mS/s to 100 MS/s.

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 8026 with in 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 4096) and different waveforms can be loaded into each segment, each having a unique length. Minimum segment size is 16 points, as long as its playback time is more than 10 μs. Information on how to partition the memory, define segment length and download waveform data to the 8026 is given in the following paragraphs.

TRACe#<header><binary_block>
Purpose
This command will download waveform data to the 8026 memory without accessing the DMA port. This command is recommended in places where DMA capability is not available from the controller. It is also the only way to download waveform data to the 8026 if you are using the RS232 link.

Waveform data is loaded to the 8026 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 14-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 4-1.
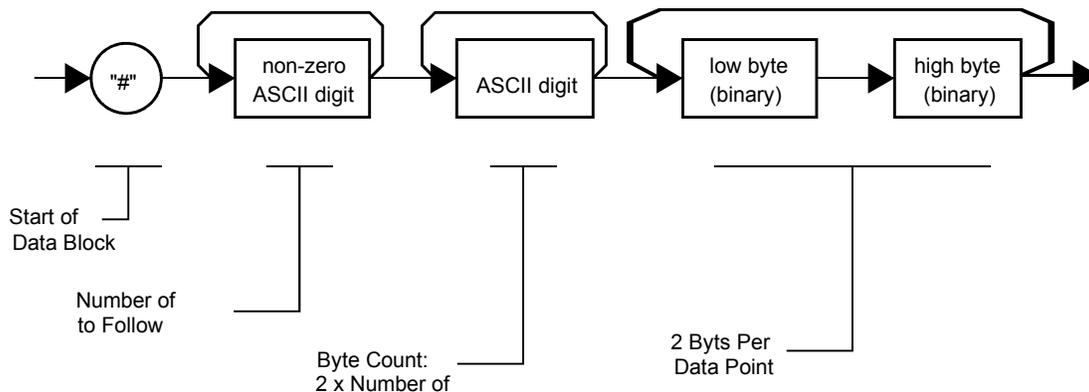

Figure 4-1. Definite Length Arbitrary Block Data Format

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, the GPIB link has 8 data bas lines and accepts 8-bit words only. Therefore, the data has to be prepared as 16-bit words and rearranged as two 8-bit words before it can be used by the 8026 as waveform data points. The following description shows you how to prepare the data for downloading to the 8026. There are a number of points you should be aware of before you start preparing the data:

1. Each channel has its own waveform memory. Therefore, make sure you selected the correct active channel before you download data to the generator
2. Waveform data points have 14-bit values
3. Data point range is 0 to 16,383 decimal
4. Data point 0 to data point 16,383 corresponds to full-scale amplitude setting. For example, if your amplitude setting is 5V$_{pk-pk}$, your generator will output waveforms from –2.5V to +2.5V. The corresponding level in waveform points is decimal 0 (0x0000) for –2.5V and decimal 16,393 (0x3FFF) for 2.5V. Similarly, the 0V point will correspond to decimal 8191 (0x1FFF)
5. The two most significant bits – D14 and D15 are control bits and are not available for normal programming. They must be set to "0" at all times except during DMA download where the last word is sent with D15 set to "1". Information on this special mode is given later in this chapter

Figure 4-2 shows how to initially prepare the 16-bit word for a waveform data point. Note that there are 14 bits used for data representation. The other two bits are used for control purpose and must be set to "0". Also note that the 8026 can not accept formats as shown in Figure 4-2; Data has to be further manipulated to a final format that the instrument can accept and process as waveform point.
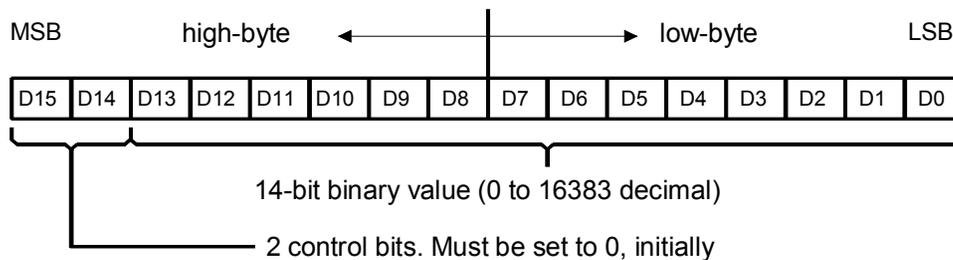
Figure 4-2. 16-bit Initial Waveform Data Point Representation

Figure 4-3 shows the same 16-bit word as in Figure 4-2, except the high and low bytes are swapped. This is the correct format that the 8026 expects as waveform point data. The first byte to be sent to the generator is the low-byte and then high-byte.
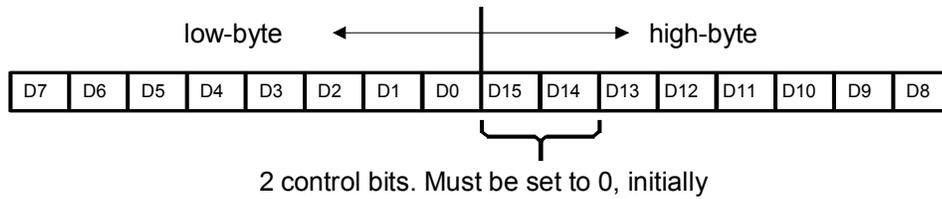
Figure 4-3. 16-bit Waveform Data Point Representation

As an example, Figure 4-4 shows word value of decimal 8025 (0x1F59) in a correct format for downloading to the 8026. The byte containing 59 is sent first and then the byte containing 1F.
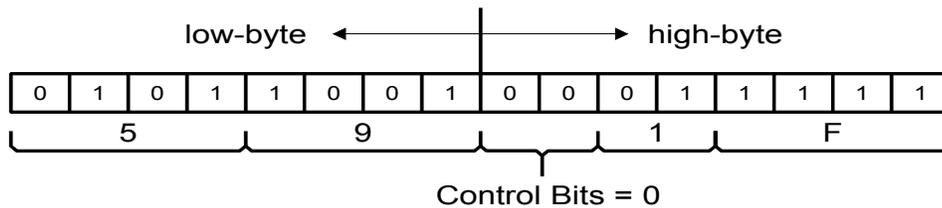
Figure 4-4. 0x1F59 Data Point Representation

TRACe:DEFine<segment_number>,<length>
Purpose
This command will define waveform segments and their relative length. Note that if you are using the TRAC:DATA# header to download waveform data, there is no need for this command because the header contains segment size information and therefore, the segment will resize automatically. The use of this command is absolutely a must if you download waveform data through the DMA port.

- <segment_number> will set the selected segment

- <length> will assign length to the above selected segment. Minimum segment length is 16 points, the maximum is limited by the memory option that is installed in your instrument either 1Meg or 4Meg.

NOTE
The 8026 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.

TRACe:DELete<segment_number>

Purpose
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.

● <segment_number> will select the segment number that will be deleted

TRACe:DELete:ALL
Purpose
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>
Purpose
This command will select 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

● <segment_number> will set the active waveform segment number

Parameter type
Numeric (integer only)

Parameter range
  <segment_number>   1 to 4096

TRACe:SELect?
Response
The 8026 will return the active segment number.

SEGment#<header>
Purpose
This command will divide 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 8026 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 two segments with 12 bytes of data that contains start address and segment size information.

```
SEGment#212<binary_block>
```

This command causes the transfer of 12 bytes of data (2 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 6.

The generator accepts binary data as 48-bit integers, which are sent in two-byte words. Therefore, the total number of bytes is always six times the number of segments. For example, 36 bytes are required to download 6 segments to the segment table. The IEEE-STD-488.2 definition of Definite Length Arbitrary Block Data format is demonstrated in Figure 4-1. 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.

- <binary_block>   Represents segment table data.

The segment table data is made of 48-bit words however, the GPIB link has 8 data bas lines and accepts 8-bit words only. Therefore, the data has to be prepared as 48-bit words and rearranged as six 8-bit words before it can be used by the 8026 as segment table data. Figure 4-5 shows how to prepare the 48-bit word for the segment start address and segment size. There are a number of points you should be aware of before you start preparing the data:

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 4096
3. Maximum segment size depends on your installed option. With the basic 8026 you can program maximum 1 Meg in one segment. With the 4 Meg option, you can use the full size of 4 Meg
4. Segment table data has 48-bit values of which 24 bits are used for start address and 24 bits are used for segment size. Therefore, Data for each segment must have 6 bytes
5. The number of bytes in a complete segment table must divide by 6. The Model 8026 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
6. First segment address starts at 256 decimal or **0x100** in hex
7. Compute the start address for segment "n" using the following equation:

$$ADD_n = ADD_{n-1} + (SIZE_{n-1} / 4)$$

For example, say you have two segments, the first is 10,000 points. The start address for segment number 2 is as follows:

$ADD_2 = ADD_1 + (SIZE_1 / 4) =$

$= 256 + 10{,}000 / 4 = 256 + 2{,}500 = 2756$; and in hex

$= 0x100 + 0x9C4 = 0xAC4$

The segment size is entered using the actual size of the segment. **Do not** divide the segment size as was done for the address. For a size of 10,000 use: 0x2710
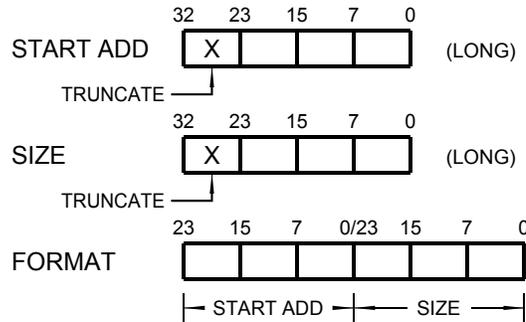
Figure 4-5. 48-bit Segment Table Download Format

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 8026 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 the section entitled **Generating Arbitrary Waveforms**.

An example of how sequenced waveforms work is demonstrated in figure 1-1a through 1-6d. The sequence generator lets you link and loop segments in user-defined order. Figure 1-6d 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 ArbExplorer program. Information how to use the ArbExplorer 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 later 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.

SEQuence#<header><binary_block>
Purpose
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

8026 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 two-step sequence with 12 bytes of data that contains segment number, repeats (loops) and mixed mode flag option.

```
SEQuence#212<binary_block>
```

This command causes the transfer of 12 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.
- "12" is the number of bytes to follow. This number must divide by 6.

The generator accepts binary data as 48-bit integers, which are sent in two-byte words. Therefore, the total number of bytes is always six times the number of sequence steps. For example, 12 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 4-1. 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.

- <binary_block>   Represents sequence table data.

The segment table data is made of 48-bit words however, the GPIB link has 8 data bas lines and accepts 8-bit words only. Therefore, the data has to be prepared as 48-bit words and rearranged as six 8-bit words before it can be used by the 8026 as sequence table data. Figure 4-6 shows how to prepare the 48-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 6. The Model 8026 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 affect 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 will be sensed at the trigger input.
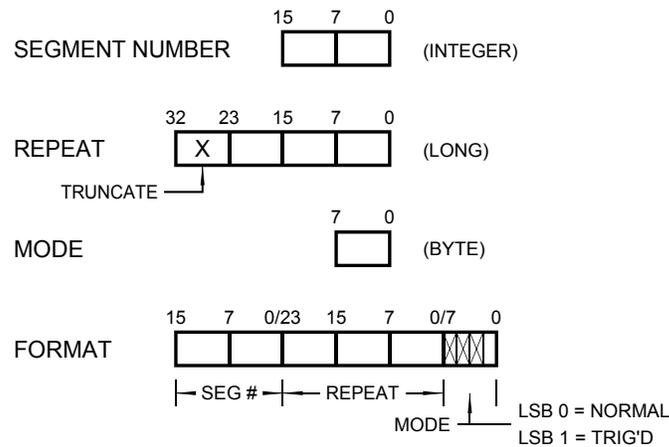
Figure 4-6. 48-bit Sequence Table Download Format

SEQuence:ADVance(?){AUTOmatic|STEP|SINGle|MIXed}
Purpose

This command will select the sequence advance mode. The way the instrument advances through the sequence links can be specified by the user.

- **AUTOmatic** - 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. AUTO is the default sequence advance mode.

- In **STEP** advance mode, the sequence is advanced to the next waveform only when a valid trigger is received. The output of the 8026 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.

- In **SINGle** 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 SING advance mode when the 8026 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.

- **MIXed** mode is a special mode, which 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 8026 must be 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.

SEQuence:ADVance?
Response and default
The 8026 will return the AUTO, STEP, SING, or MIX depending on the present sequence advance mode setting. Default value is AUTO

SEQuence:ADVance:SOURce(?){EXTernal|INTernal}
Purpose
This command will select the sequence advance source. The source of the event causing sequence advance can be specified by the user.

- <EXTernal> will set the external advance mode
- <INTernal> will set the internal advance mode

SEQuence:ADVance:SOURce?
Response and default
The 8026 will return the EXT, or INT depending on the present single-sequence advance source setting. Default value is EXT.

SEQuence:DEFine <step_#>,<segment_#>,<#_repeat>,<mode>
Purpose
This command will define sequence steps and their associated advance mode.

- <step_#> will define the link number
- <segment_#> will select the segment for the above link
- <#_repeat> will define how many times the selected segment will loop
- <mode> will assign the mixed mode bits for this sequence step. "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. Note that the <mode> parameter has no affect when the sequence advance mode is set to SING.

To understand better the way the <mode> switch affects a sequence, consider the following sequence setting:

```
SEQ:DEF 1,1,1,0        (Step 1 = segment 1, no loops, mode 0)
SEQ:DEF 2,2,1,1        (Step 2 = segment 2, no loops, mode 1)
SEQ:DEF 3,3,10,1       (Step 3 = segment 3, 10 loops, mode 1)
SEQ:DEF 4,4,1,0        (Step 4 = segment 4, no loops, mode 0)
```

After selecting the sequence mode, the generator will generate segment 1 and automatically advance to segment 2. It will continue to generate segment 2 until a valid trigger is received. After it has received a trigger, the generator will complete segment 2 and advance to segment 3. It will continue to generate and loop segment 2 until a valid trigger is received. After it has received a trigger, the generator will complete 10

loops of segment 3 and will advance to segment 4. From segment 4 the generator will automatically advance to the beginning of the sequence and will repeat the above.

NOTE

Although trigger signals are used to advance mixed mode, the mixed mode operates in continuous mode only. The <mode> parameter will be ignored if you will use SING as advance mode for the sequence table.

TIP

Every time you use the SEQ:DEF command while your 8026 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:ALL
Purpose
This command will erase the sequence table.

DMA{OFF|ON|0|1}
Purpose
This command will set the DMA (Direct Memory Access) state on and off.

NOTES
1. **DMA mode is not available if you are using RS232**
2. **Before you attempt using DMA data transfer, make sure that your GPIB controller has DMA capability**
3. **When downloading waveforms using the DMA, only binary data can be loaded. The binary data structure is shown in Figure 4-1. Be careful not to send ASCII characters or SCPI commands that are not part of the waveform data because it will confuse the generator and will cause errors.**

- ON or 1 will set the 8026 in DMA state. After this, the instrument cannot accept normal commands but only binary bytes. Normal command mode is resumed when the DMA state is terminated. The DMA state is terminated automatically to DMA OFF when the most significant bit of the high byte is set to "1". Therefore, your last waveform point must contain DMA termination code as shown in figure 4-7.

- OFF or 0 is the default DMA setting. Sending this command will have no effect on the generator and will not generate error messages even when the DMA is set to ON.

DMA:TYPE(WAVE|FM)
Purpose
This command will set the DMA port to accept waveform data, or FM modulating waveform data.

- WAVE will select the DMA for downloading arbitrary data to the waveform memory
- FM will select the DMA for downloading arbitrary data to the FM modulating waveform memory

Downloading Waveforms Using the DMA Port

When selecting the WAVE option, the DMA mode prepares the 8026 to accept waveform data directly from the controller's DMA port, without traffic control of the internal CPU and without standard GPIB handshake. Therefore, DMA data transfer is the fastest way to get waveforms to the generator. When set to this mode, the internal CPU disconnects from the waveform memory and passes control to a special DMA controller, the data bus is connected directly to the GPIB data bus, and data is downloaded into the memory in binary blocks. Bytes and bit order are the same as for the Arbitrary Block transfers as shown in Figures 4-1, 4-2 and 4-3. After the data is loaded into the 8026, control is returned to the instrument.

NOTES
1. **DMA mode is not available if you are using RS232**
2. **Before you attempt using DMA data transfer, make sure that your GPIB controller has DMA capability**
3. **When downloading waveforms using the DMA, only binary data can be loaded. The binary data structure is shown in Figure 4-1. Be careful not to send ASCII characters or SCPI commands that are not part of the waveform data because it will confuse the generator and will cause errors.**

Before you write to a segment, you must first define it using the TRACe:DEFine command. You can define more than one segment in preparation for downloading waveforms however, before you start downloading waveforms, you first must select the active segment using the TRACe:SELect command.

- ON or 1 will set the 8026 in DMA state. After this, the instrument cannot accept normal commands but only binary bytes. Normal command mode is resumed when the DMA state is terminated. The DMA state is terminated automatically to DMA OFF when the most significant bit of the high byte is set to "1". Therefore, your last waveform point must contain DMA termination code as shown in figure 4-7.

- OFF or 0 is the default DMA setting. Sending this command will have no effect on the generator and will not generate error messages even when the DMA is set to ON.

In summary, the following sequence should be used for DMA data transfers.

1. Controller sends commands:

```
TRAC:DEF <segment_number>,<length>
*OPC? (wait for 8026 response "1")
TRAC:SEL <segment_number>
DMA ON
```

2. Apply 100 ms delay to allow stability of the DMA port

3. Use an I/O routine such as ViMoveAsync (from the VISA I/O library) to transfer binary blocks of data to the generator.

4. Last waveform point is sent with high byte's MSB bit set to "1"; DMA transfer is completed and CPU resumes control.

As an example, Figure 4-7 shows word value of decimal 8025 (0x1F59) in a correct format for terminating the DMA mode.
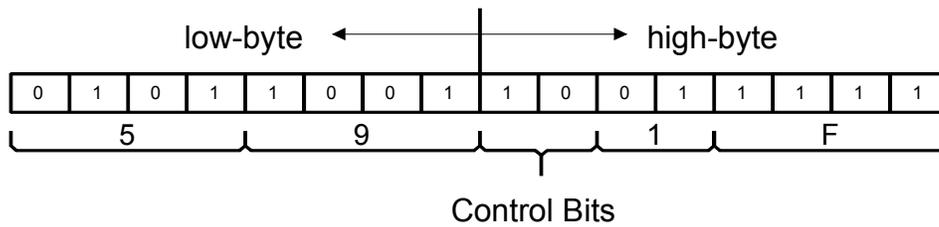
Figure 4-7. Terminating the DMA mode with the MSB Control bit set to "1"

Downloading FM Waveforms Using the DMA port

When selecting the FM option, the DMA mode prepares the 8026 to accept FM modulating waveform data directly from the controller's DMA port, without traffic control of the internal CPU and without standard GPIB handshake. Therefore, DMA data transfer is the fastest way to get modulating waveforms to the generator. When set to this mode, the internal CPU disconnects from the waveform memory and passes control to a special DMA controller, the data bus is connected directly to the GPIB data bus, and data is downloaded into the memory in binary blocks. After the data is loaded into the 8026, control is returned to the instrument.

Downloading data to the 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.

NOTES
1. **DMA mode is not available if you are using RS232**
2. **Before you attempt using DMA data transfer, make sure that your GPIB controller has DMA capability**
3. **When downloading waveforms using the DMA, only binary data can be loaded. The binary data structure is shown in Figure 4-8. Be careful not to send ASCII characters or SCPI commands that are not part of the waveform data because it will confuse the generator and will cause errors.**

The following sequence should be used for FM Waveform with DMA through the data port:

1. Prepare your waveform data using the following relationship:

    N = SCLK [Hz] x 14.31655765

2. Convert the result from 1 above to 4-byte (32-bit) binary word and union with an 8-bit control word as shown in Figure 4-8.

3. Controller sends commands:

    ```
    DMA:TYPE FM
    DMA ON
    ```

4. Use an I/O routine such as ViMoveAsync (from the VISA I/O library) to transfer binary blocks of data to the generator.

5. Last data point is sent with high byte's MSB bit set to "1"; DMA transfer is completed and CPU resumes control.

The FM modulating waveform data is made of 40-bit words. However, the GPIB link has 8 data bas lines and accepts 8-bit words only. Therefore, the data has to be prepared as 40-bit words and rearranged as

five 8-bit words before it can be used by the 8026 as FM modulating waveform data. Figure 4-8 shows how to prepare the 40-bit word for the FM modulating waveform.

There are a number of points you should be aware of before you start preparing the data:

1. The FM function is shared by both channels
2. The number of bytes in a complete FM modulating waveform data must divide by 5. The Model 8026 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
3. The LSBit on the last byte sets marker position. "0" = sets no marker and "1" sets marker. You can set as many markers as you want.
4. The SYNC output serves as marker output when you have the 8026 set to operate in FM mode. Normal SYNC level is TTL low. The SYNC output is set to TTL high when the marker bit is true. This way you can use the SYNC output to mark frequency occurrences during FM operation.
5. DMA download is terminated with the MSBit of the last byte set to 1.

As an example, lets compute the number that has to be downloaded for a sample clock value of 15.23456MHz. Using the equation $N = SCLK [Hz] \times 14.31655765$, we compute

$N = 15.23456MHz \times 14.31655765 = 218106456.5$, rounded to $218106457 = 0x0D000A58$
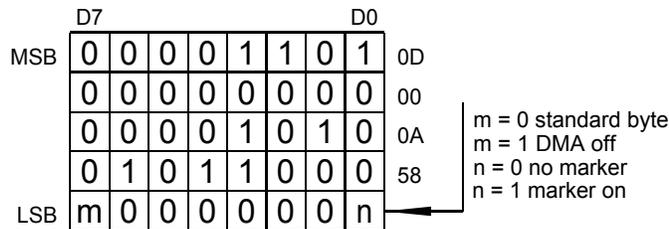
Figure 4-8. FM Modulating Waveform Data Strructure

The bytes are downloaded to the DMA port byte at a time. The first byte is the MSByte and the last is the LSByte. The LSByte is a control byte and is interpreted as follows:

0x00 – normal data byte, no marker bit

0x01 – normal data byte, marker bit on

0x80 – DMA termination byte, no marker bit

0x81 – DMA termination byte, marker bit on

DMA:TYPE?
Response and default
The 8026 will return WAVE, or FM depending on present setting. Default value is WAVE.

SYSTem Subsystem

The system-related commands are not related directly to waveform generation but are an important part of operating the 8026. These commands can reset or test the instrument, or query the instrument for system information.

Keyword	Parameter Form	Default,Low Limit,High Limit
:SYSTem		
:ERRor?		
:VERSion?		
:SAVE	\<memory#\>	**1**,1,10
:RECall	\<memory#\>	**1**,1,10
RESet		
*RST		
TEST?		
*TST?		
*IDN?		

SYSTem:ERRor?
Purpose
This query will interrogate the 8026 for programming errors.

Response to query
The 8026 will return error code. Error messages are listed later in this manual.

SYSTem:VERSion?
Purpose
This query will interrogate the 8026 for SCPI version.

Response to query
The 8026 will return 1999.0.

SYSTem:SAVe \<memory#)
Purpose
This command will save the entire front panel setting. This command will not save arbitrary memory and segment partitions. It will also not save sequence table data.

● \<memory#\>   will select the memory cell where front panel settings will be stored.

Parameter type and range
Integers from 1 through 10

SYSTem:RECall \<memory#)
Purpose
This command will recall and update front panel setting from parameters stored with the SYST:SAVe command. This command will not restore arbitrary memory and segment partitions. It will also not restore sequence table data.

● <memory#>   will select the memory cell where front panel settings will be recalled from.

Parameter type and range
Integers from 1 through 10

RESet, *RST
Purpose
This command will reset the 8026 to factory defaults.

TEST?, *TST?
Purpose
This query will invoke the 8026 selftest function.

Response to query
The 8026 will return "0" for no error, or error code. Error codes are listed below.

Error Code	Description
0	No errors, selftest passed
1	CPU and peripherals communication failure
2	Channel 1 failure
4	Channel 2 failure
8	Sequence generator channel 1 failure
16	Sequence generator channel 2 failure
32	Channel 2 sample clock divider failure

*IDN?
Purpose
This query will interrogate the identity of the 8026.

Response to query
The 8026 will return Tabor Electronics,8026,0,1.0.

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 has specified 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.

The following is a complete listing of all common-commands and queries, which are used by the 8026

*CLS - Clear the Status Byte summary register and all event registers.

*ESE <enable_value> - Enable bits in the Standard Event enable register. The selected bits are then reported to the status byte.

*ESE? - Query the Standard Event enable register. The generator returns a decimal value, which corresponds to the binary-weighted sum of all bits, set in the register.

*ESR? - Query the Standard Event register. The generator returns a decimal value, which corresponds to the binary-weighted sum of all bits, set in the register.

*IDN? - Query the generator's identity. The returned data is 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, Model 8026 response to *IDN? is:

Tabor Electronics,8026,0,1.0

*OPC - Set the "operation complete" bit (bit 0) in the Standard Event register after the previous commands have been executed.

*OPC? - Returns "1" to the output buffer after all the previous commands have been executed. *OPC? is used for synchronization 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.

*OPT? - Returns the value "0" for a 8026 with no options. Returns "1" for a Model 8026 with 4Meg memory.

*RST - Resets the generator to its default state. Default values are listed in Table 4-1.

*SRE <enable_value> - Enables bits in the Status Byte enable register.

*SRE? - Query the Status Byte enable register. 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? - Query the Status Byte summary register. The *STB? command is similar to a serial poll but is processed like any other instrument command. The *STB? command returns the same result as a serial poll, but the "request service" bit (bit 6) is not cleared if a serial poll has occurred.

*TRG - Triggers the generator from the remote interface. This command effects the generator if it is first placed in the Trigger or Burst mode of operation and the trigger source is set to "BUS".

*TST? - Implements an internal self-test and returns a value as described below. Approximately 90% of the 8026 functionality is tested.

       0 - Selftest passed

       1 - CPU to peripherals communication failure

       2 – Channel 1 failure

       4 – Channel 2 failure

       8 – Sequence generator failure

More than one failure can be reported at one time. For example, the returned value "17" indicates both a TBD problem and a TBD problem exist.

***WAI** – Wait for all pending operations to complete before executing any additional commands over the interface.

The SCPI Status Registers

The Model 8026 uses the Status Byte register group and the Standard Event register group to record various instrument conditions. Figure 4-1 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.

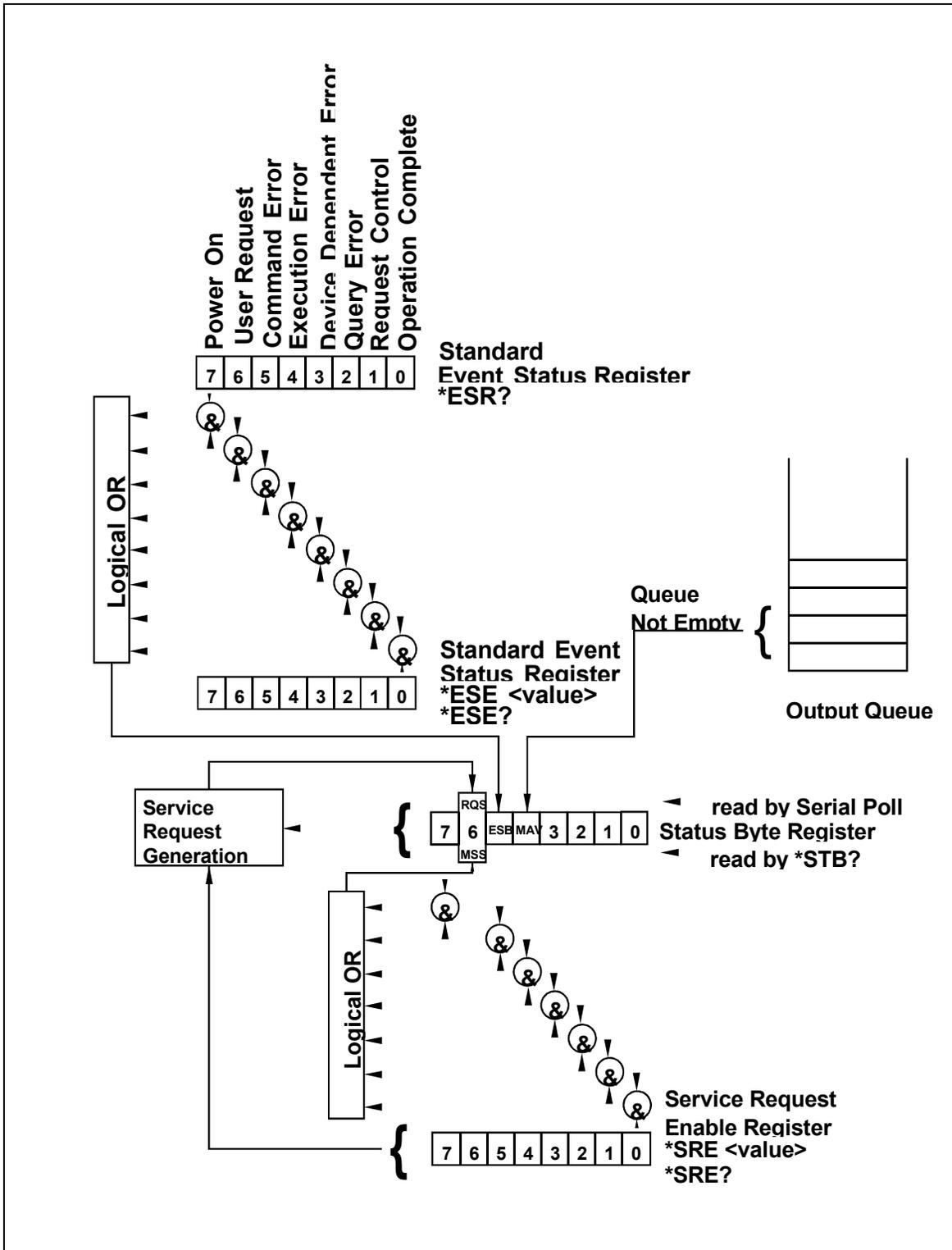Bit 7 - Decimal value 128. Not used, always set to 0.

Figure 4-9. SCPI Status Registers

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.

*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 Model 8026.

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 8026 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. Corrective action: Change parameters to correct the problem.

2. ampl/2 + |offset| is more than 5. Corrective action: Reduce offset to 0, then change amplitude-offset values to correct the problem.

3. Activating filters when the 8026 is set to output the built-in sine waveform, or activating the built-in sine waveform when one of the 8026 filters is turned on. Corrective action: If in sine, select another function and activate the filter(s).

4. Activating burst mode when the 8026 is set to sequence mode, or activating sequence mode when the 8026 is set to burst mode. Corrective action: Remove the 8026 from burst or sequence and then selected the desired mode.

5. Changing operating mode from triggered to continuous when the 8026 is set to single sequence advance, or changing the operating mode from continuous to triggered when the 8026 is set to automatic sequence advance mode. Corrective action: Observe the 8026 advance mode while setting sequence advance.

-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.

Chapter 5

ArbExplorer©

What's In This Chapter

This Chapter contains information how to install, invoke and use ArbExplorer. Introduction to ArbExplorer and examples how to program instrument controls and parameters and how to generate waveforms and download them to the 8026 are also given in the following sections.

Introduction To ArbExplorer

In general, ArbExplorer is a utility program that serves as an aid for programming the Model 8026. ArbExplorer has many functions and features of which all of them share a common purpose – controlling 8026 functions from remote. As minimum, to use ArbExplorer, you'll need the following tools:

1. Computer, Pentium II or better

2. Windows 95, or higher

3. High resolution screen, at least, 1024 x 768 pixels

4. Pointing device, mouse or ball

5. GPIB interface card, National Instruments TNT488.2

6. Visa 2.0, or higher installation

7. Last, but not least, some basic knowledge how to operate computers and Windows-based programs.

ArbExplorer operation is divided into three main functions: 1) Front panel control, 2) Waveform generation and editing and 3) FM waveform generation and editing. These operating options are described in this chapter. But, you must install ArbExplorer before you can use it, so the next paragraphs describe installation and first steps before going into in-depth operation.

Installing ArbExplorer

The installation program installs ArbExplorer on a logical drive of your choice. The default is drive C. It automatically creates a new directory and copies the files that are required to run the program. Before you install ArbExplorer, make sure that there is at least 5 megabytes of available memory on your hard disk drive.

To install ArbExplorer, insert the distribution disk in the A: drive.

Invoke Run and type:

```
A:\Setup
```

The install program does the complete job far you and creates a workgroup and icons to start ArbExplorer.

Quitting ArbExplorer

Before you start roaming through menus and editing commands, we strongly recommend that you make yourself familiar with ArbExplorer basics and concept. For now quit the program and spend some more time with this section of the manual. Point the mouse cursor to the File menu and press the left mouse button. Move the mouse cursor to the Exit command and press the left mouse button.

For the New and Advanced Users

For the New User
Learning to use ArbExplorer is easy, intuitive and quick, even if you have never used such programs before. After you have installed ArbExplorer on your computer read the following paragraphs to learn how to find your way around ArbExplorer's menus.

Once you are familiar with the basics, you'll continue to learn about features, programming, and editing commands. If you can't find the answer to a question in this guide, call your distributor or the Tabor customer support service near you and we'll gladly assist you with your problems.

For the Advanced User
If you are already familiar with computer conventions and have basic knowledge of Windows programming, you may want to skip some of the following paragraphs.

Conventions Used in This Manual

This manual uses certain typographical conventions to make it easier for you to follow instructions. These conventions are described in the following:

[Enter, or ↵] Press the Enter or Return key.

[Esc] Press the Escape key.

[Alt-F] Press the Alt key and the key that follows, simultaneously. In

this example the key that follows is F.

[Ctrl-S] Press the Control key and the letter that follows, simultaneously. In this example, the letter is S. The control key also appears in the menus as a target sign.

[↑] [↓] [→] [←] Press the Arrow key with the symbol pointing in the direction specified (i.e., up, down, left, or right).

<+> Press the key for the character or word enclosed in angle brackets. In this case, the Plus sign key.

The Opening Screen

Invoke ArbExplorer by double clicking on the icon. If you cannot find the icon on your desktop, click on Start, Programs and ArbExplorer. The opening screen will show. If you installed the program correctly, your screen should look as shown in Figure 5-1.
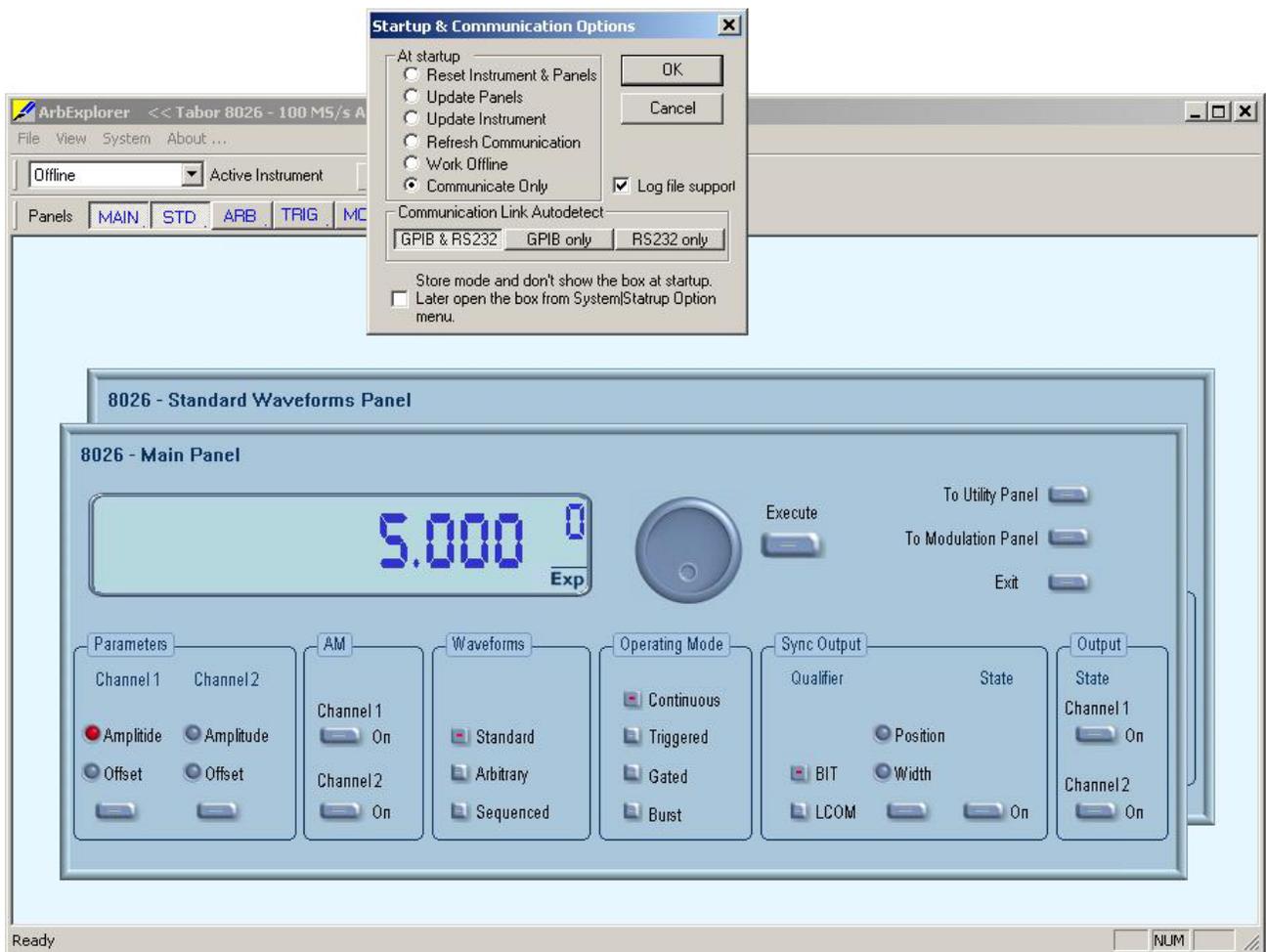
Figure 5-1. The opening Screen

The Startup & Communication Options dialog box is displayed at the top. You can check the "Store and don't show…" so next time you invoke ArbExplorer, this dialog box will not be displayed. The purpose of this dialog box is to update the program in the way you intend to use it. For example, if you are using the GPIB link and do not intend to use RS232, then you can click on the GPIB only button and check the Communicate only option so the next time you use ArbExplorer, the program will automatically resume GPIB communication with the same address as was originally detected.

If you chose to hide this dialog box, you can still access and change the options from the System command, at the top of the screen.

Make your selection and click OK. The Startup & Communication Updater dialog box will be removed from the screen. And two panels: the Main and the standard Waveform panel will now be accessible. But before we go into panel operation, lets look at the toolbars at the left top of the screen as shown in Figure 5-2.

Figure 5-2. ArbExplorer's Toolbars

The standard Windows **Menu Bar** is the top bar. It provides access to main system controls like saving files, and viewing or removal of screen images.

The second bar is called **Active Instrument** bar. It provides direct access to different instruments that are active on the GPIB, or RS232 links. ArbExplorer can operate five 8026 units simultaneously. If the instruments were connected to the GPIB link while invoking ArbExplorer, they will automatically be detected by the program and will be assigned numbers from 1 through 5. Buttons that were not attached to an instrument will stay gray. The active instrument is displayed with its associated address. For example, with two instruments having addresses 11 and 12, pressing button "1" will bring up GPIB address 11 and pressing "2" will bring up GPIB address 12. You can also select the active instrument from the GPIB address window.

HINTS

---

1. **Active instrument means that changes you make on panels and waveform composers will affect this instrument only.**

---

2. **ArbExplorer can address only one RS232 instrument.**

---

3. **ArbExplorer can address any combination of GPIB and RS232 instruments as long as there is only one RS232 instrument attached to the computer.**

The **Panels** bar provides direct access to instrument control panels. The individual control panels are explained later in this chapter. The MAIN, STD, ARB, TRIG, MOD and UTIL buttons will bring up to the screen panels that are associated with these names. The WAVE and FM buttons will open the waveform and FM waveform composers. The opening screen has two panels – MAIN and STD so the two buttons on the Panels bar are depressed. Click on other buttons and interactively get the feel how ArbExplorer opens and cascades panels.

ArbExplorer Features

In general, ArbExplorer has one basic purpose – controlling 8026 functions and parameters. On the other hand, the 8026 has three main features: 1) Generating standard waveforms, 2) Generating arbitrary waveforms and 3) Generating modulated waveforms. So, ArbExplorer had to be constructed in such a way to provide complete control capability from the computer screen.

ArbExplorer has three main screens: 1) Control panels, 2) Waveform composer and 3) FM waveform composer. The various screens along with instructions how to access and use them are described below in detail.

The Control Panels

The control panels look and feel just as if you would operate an instrument from its front panel. They even look like a front panel of an instrument, so operating function and changing parameters are very intuitive. Lets look at the first panel that shows at the opening screen. This panel, as shown in Figure 5-3, is called the Main Panel.

To begin with, lets explore the panel controls to see how they feel, react and what they do. All other panels share almost the same feel, so the description of how to operate the Main Panel can serve as general guide for controlling the rest of the panels.

Looking at the panel you can identify the following controls: Push buttons, LED's, radio buttons, Dial and Digital display. The function of each control is described below.

Push Buttons – These are used for two functions: a) For selecting an active parameter and b) for toggling a function on and off. For example, the Channel 1 button in the Parameters group will select between the Amplitude and Offset LED's. The selection buttons have momentary push action, so pressing (clicking a mouse on) this button will change the reading on the display to correspond with the selected parameter. The Channel 1 State button in the Output group has toggles the output on and off. The first mouse click will push the button inwards and will turn on a red bar at the center of the button, indicating that the function is on. The second mouse click will turn the function off.

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, indicating its state condition. Pressing some radio buttons will pop up a new panel. For example, if you press the Triggered button in the operating Mode group, triggered operation will be enabled and the Trigger Panel will automatically pop up and cascaded in front of the Main Panel.

LED's – The LED's indicate which of the parameters are displayed on the Digital Display. Red LED indicates that the parameter name next to this LED is selected. Only one LED can be ON at a time.

HINT

LED's may be turned on by pressing the button below. You cannot turn LED's ON by clicking on the LED itself!

Dial – Use the dial to modify displayed reading. To use the dial, press and hold the mouse cursor on the dial and move the mouse in a clockwise circle to increase the number, or counterclockwise circle to decrease the displayed number. The dial modifies digits at the cursor position and will allow modification within the legal range of the displayed parameter. If you reach the end of the range, the dial will have no further effect on the display. If you do not want to use the dial, you can still change the display reading by using the [↑], or [↓] keys, or simply type the required number using the your standard keyboard features.

NOTE

After you change the displayed readout, the 8026 will be updated with the new parameter only after you press the Execute button.

Digital Display – The display is used for displaying and reading various 8026 parameters, just as you would use it on your instrument.

The Main Panel

The Main Panel, as shown in Figure 5-3, is the first panel you see after invoking ArbExplorer. If you press the Exit button on the main panel, you'll be exiting the program completely. Notice how buttons and LED's are grouped; this is done specifically so that common parameters are placed in functional groups. The Main Panel groups allow (from left to right) adjustment of amplitude and offset, turning ON the amplitude modulation function, selecting the waveform type, selecting the operating mode, controlling the SYNC output and turning ON, or OFF the main outputs. Controls, where applicable, are provided for each channel separately. Parameters that are associated with the second channel will be ignored, If you have Model 8025.

Figure 5-3. The Main Panel

If you are connected properly to a GPIB or RS232 link and ArbExplorer has detected this link and placed the address on its Active Instrument bar, then every time you press a button, you are getting an immediate action on the 8026. It is different if you are changing parameters on the display; Doing this, you'll have to press the Execute button for the command to update the instrument.

HINT

If your instrument has not been detected automatically, or if you connected the 8026 only after you invoked the program, you can still detect your instrument by selecting the Refresh Communication from the System command. You can always work offline if you just want to explorer the program.

The functional groups in the Main Panel are explained below.

Parameters The Parameters group contains two parameters for each channel: Amplitude and Offset. To access the required parameter, click on the button below the Channel1 or Channel 2 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, press Execute to update the 8026 with the new reading.

Note

Normal color of the digital reading is dark blue. If you modify the reading, the color changes to a lighter shade of blue, indicating that the 8026 has not been updated yet with the new parameter. Pressing Execute will update the instrument and will restore the color of the digital readout to dark blue, indicating that the displayed value is the same as the generator setting.

Also note that the digital readout has an autodetect mechanism for the high and low limits. You cannot exceed the limits if you are using the dial but only if you use the keypad. In case you do, the program will not let you download an illegal parameter and you'll be requested to correct your setting.

AM The buttons in the AM group let you turn on the AM function. There is a button for each channel. Click on the Channel 1 button to turn AM ON for channel 1 and click on the Channel 2 button to turn AM ON for channel 2. Note that ON is indicated by a red bar at the center of the button.

Waveform The Waveform group is used for selecting between waveform types. The 8026 provides three types of waveforms: Standard, Arbitrary and Sequenced. Pressing one of these buttons will cause two reactions: 1) The 8026 will change the output waveform to the selected option and 2) Another panel, associated with the selected waveform type will be invoked and brought to the front. The default waveform type is Standard. Therefore, the opening screen has this panel behind the Main Panel. If you want to change standard waveform parameters, you can either click on the standard waveform panel, or select this panel from the Panels bar.

Operating Modes The Operating Modes group is used for selecting the active trigger mode for the instrument. You can select between continuous, triggered, gated and burst modes. There is no additional panel associated with the continuous mode, but if you press one of the other operating mode options, the 8026 will be updated with the new trigger

option and the Trigger Panel will pop up, allowing adjustments of trigger parameters.

SYNC Output

The SYNC Output group controls SYNC parameters and enables or disables the SYNC output. The SYNC output is enabled when the State button is ON. The SYNC Width and Position parameters will affect the 8026 when the BIT validator is selected. In LCOM mode, the width and the position will have no affect on the SYNC output.

Output

The Output group is used for controlling the state of the 8026 channel 1 and 2 outputs. The buttons toggle the output ON and OFF. The Channel 1 output will be ON when you click on the Channel 1 button and the red bar at the center illuminates.

Besides the grouped buttons and parameters, there are three other buttons on the right top corner of the panel. These buttons will provide access to the (from top) Utility and Modulation Panels. Also, if you press the Exit button, the program will be terminated.

The Standard Waveforms Panel

The Standard Waveforms panel, as shown in Figure 5-4, is always invoked at the back of the Main Panel since the default output shape of the 8026 is standard sine waveform. There are a number of ways to access this panel: You may click on it to bring it in front of the main Panel, you can click on the STD. Button on the Instruments bar, or you can click on the Standard button within the Waveforms group in the Main Panel. The functional groups in the Standard Waveforms Panel are described below.

Channel 1&2

The Channel 1 and Channel 2 Parameters group lets you select the parameters for the standard waveforms. There are nine standard wave shapes that the 8026 can generate and by selecting one of the buttons, you automatically select this function and provide access to parameters that are unique for this function. For example, if you click on the Square button, the Duty Cycle parameter will be available for modification. Likewise, if you select the shape, the fall, high, rise and delay parameters will be available for modification. Note that by clicking a button in this group, you are immediately updating the 8026 output with this waveform shape.

Figure 5-4. The Standard Waveforms Panel

10 MHz Ref   The 10 MHz Ref group provides selection between internal or external references. The 8026 has reference of 1ppm TCXO as Standard An external reference input is provided for applications requiring better accuracy and stability. Click on the Internal button to select the Internal reference, or the External button to activate the external reference input.

WARNING

By selecting an external reference you are disabling the internal reference circuit. If you do not have a 10 MHz reference connected to the instrument, the output will generate erroneous frequencies.

Frequency   The button in the Frequency group lets you program the output frequency of the selected waveform shape. The frequency parameter may be modified when the LED illuminates. You can use the dial, keyboard, or the [↑] [↓} keys to adjust the readout to the required setting. After you modify the reading, press Execute to update the 8026 with the new reading.

Note

Normal color of the digital reading is dark blue. If you modify the reading, the color changes to a lighter shade of blue, indicating that the 8026 has not been updated yet with the new parameter. Pressing Execute will update the instrument and will restore the color of the digital readout to dark blue, indicating that the displayed value is the same as the generator setting.

> **Also note that the digital readout has an autodetect mechanism for the high and low limits. You cannot exceed the limits if you are using the dial but only if you use the keypad. In case you do, the program will not let you download an illegal parameter and you'll be requested to correct your setting.**

Besides the grouped buttons and parameters, there are two other buttons on the right top corner of this panel. These buttons will provide back access to the Main Panel and will also allow removal of the Waveforms Panel from the screen.

The Arbitrary & Sequence Panel

The Arbitrary & Sequence panel, as shown in Figure 5-5, is invoked either by pressing the ARB button on the Panels bar, or clicking on the Arbitrary or Sequenced Waveforms on the Main Panel. Note that if you invoke the Arbitrary & Sequence Panel from the Panels menu, the 8026 will not change its output type. On the other hand, if you select the arbitrary, or the sequenced options from the Main Panel, besides invoking the Arbitrary & Sequenced Panel, the 8026 will immediately change its output type to the selected waveform type. The functional groups in the Arbitrary Waveforms Panel are described below.

Parameters

The Parameters group contains two parameters for each channel: Amplitude and Offset. Actually, the values exhibited in this group are exactly the same as in the Main Panel, so every time you change amplitude and offset in the Parameters group, the other panels are updated automatically.

To access the required parameter, click on the button below the Channele1 or Channel 2 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, press Execute to update the 8026 with the new reading.

Note

Normal color of the digital reading is dark blue. If you modify the reading, the color changes to a lighter shade of blue, indicating that the 8026 has not been updated yet with the new parameter. Pressing Execute will update the instrument and will restore the color of the digital readout to dark blue, indicating that the displayed value is the same as the generator setting.

Also note that the digital readout has an autodetect

> **mechanism for the high and low limits. You cannot exceed the limits if you are using the dial but only if you use the keypad. In case you do, the program will not let you download an illegal parameter and you'll be requested to correct your setting.**

Figure 5-5. The Arbitrary & Sequence Panel

Sample Clock   The Sample Clock group is comprised of parameters that control the sample clock. Select the SCLK LED to adjust the sample clock rate and the Divider (2) LED to adjust the sample clock divider for channel 2. Note that the sample clock rate is programmed in units of S/s (samples per second) and will affect the Model 8026 only when programmed to output arbitrary or sequenced waveforms.

To access the required parameter, click on the button 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, press Execute to update the 8026 with the new reading.

The Source buttons provide selection between the internal sample clock generator or, if you want to synchronize your instrument to an external clock, you can select the External option. If you select the external source option, you must connect an external source to your instrument, otherwise you will have no output at all.

10 MHz Ref   The 10 MHz Ref group is a duplicate of the same group in the Main Panel. The 10 MHz Ref group provides selection between internal or external references. The 8026 reference is 1ppm TCXO as Standard.

An external reference input is provided for applications requiring better accuracy and stability. Click on the Internal button to select the Internal reference, or the External button to activate the external reference input.

WARNING

By selecting an external reference you are disabling the internal reference circuit. If you do not have a 10 MHz reference connected to the instrument, the output will generate erroneous frequencies.

Sequence Advance   The Sequence Advance group provides control over advance modes for the sequence generator. Advance options are: Auto, Stepped, Single and Mixed. Refer to the 8026 manual to find out more when and how to use these advance modes. You should be careful while selecting modes because it is possible to cause settings conflict, for example, if you select the Single option and you forgot to change your trigger mode to Triggered.

The External Internal switch selects the trigger advance source. External - enables the TRIG IN connector at the back of the instrument. Internal – disables the trigger input and routes an internal trigger generator to the trigger advance circuit.

Besides the grouped controls, there are five other buttons available on the Arbitrary & Sequence Panel. These buttons provide access to (from top to bottom) Wave Composer, Sequence Table, Segment Table, Main Panel. The last button removes the Arbitrary & Sequence Panel from the screen. The Wave Composer will be discussed separately in this chapter. The Segment and Sequence Tables are described below.

Using the Segment Table

If you want to learn more about segment control, you should refer to section 3 of this manual. In general, the 8026 can generate arbitrary waveforms but before it can generate anything, the waveforms have to be downloaded to the instrument from a host computer. The waveforms are downloaded as coordinates and are stored in the 8026 is a place designated as "waveform memory". The waveform memory has a finite size of 1Meg, or 4Meg if you purchased the instrument with this option.

Having such a long memory to store waveforms does not necessarily mean that you have to use the entire memory every time you download a waveform. On the contrary, the 8026 allows segmentation of the memory so that up to 4096 smaller waveforms could be stored in the waveform memory. There are two ways to divide the waveform memory to segments: Define a segment and load it with waveform data, define the next and load with data, then the third etc. The second alternative is to use what ArbExplorer has to offer and that is

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

Want to use it? Here is how it is done.

Click with your mouse on the To Segment Table button. A dialog box as shown in Figure 5-6 will pop up.
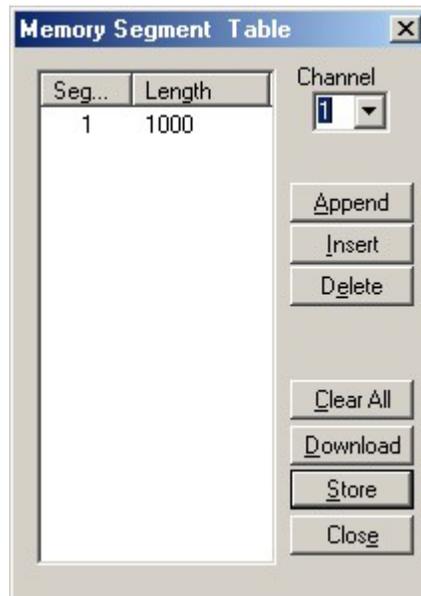
Figure 5-6. The Segment Table

The **Channel** field lets you select the active channel. If "1" is displayed, then the table you'll be constructing will affect channel 1. Likewise, select "2" if you want to program the table for channel 2.

The two main fields in the segment table are Seg and Length. The **Seg** (segment) is an index field, which can have values only, from 1 to 4096. The **Length** is always associated with the segment number. You can program any segment length from 16 to 4 Meg provided that the sum of the length for all programmed segments does not exceed your purchased waveform memory size.

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

The **Clear All** key will remove all segments from the table and will let you start designing your segment table from fresh.

you start designing your segment table from fresh.

Click on the **Close** to discard of the contents of the dialog box without saving your last actions and to remove the Segment Table from the screen but click on the **Store** 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 segment table to the instrument and will store the contents of your segment table.

Using the Sequence Table

If you want to learn more about sequences and sequence control, you should refer to section 3 of this manual. In general, the 8026 can generate sequenced waveforms but waveforms must to be loaded to the instrument from a computer before it can generate sequences. The waveforms are downloaded as coordinates and are stored in the 8026 is a place designated as "waveform memory". The waveform memory has a finite size of 1Meg, or 4Meg if you purchased the instrument with this option.

As was explained in Using the Segment Table above, the waveform memory can be divided into smaller segments and up to 4096 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 loosing valuable memory space and without scarifying waveform coherences, or integrity. The tool for using repetitive and multiple segments in one long waveform is called Sequence Generator. The 8026 has two separate sequence generators, one for each channel and ArbExplorer has a special dialog box where sequences are designed. This tool is called – Sequence Table.

The Sequence Table is demonstrated in Figure 5-7. To access this table click on the "To Sequence Table" button. Description of the various controls in this table is given below.
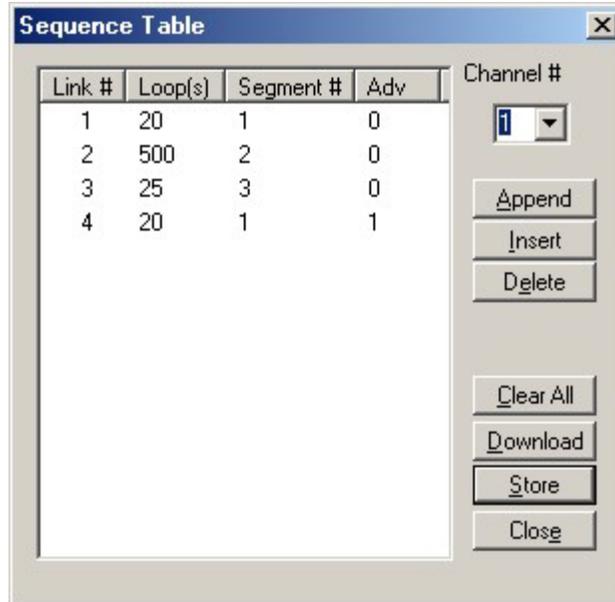
Figure 5-7. The Sequence Table

There are four major elements that you should consider while programming a sequence table. They are: Link#, Loop(s), Segment# and Adv. These elements are explained below.

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

Loop(s) – 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.

Segment # - 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 how you associate links to segments, except you cannot program in the sequence table segments that were not defined earlier.

Adv – This field is a special code that is used in conjunction with the mixed advance mode. This bit flags the 8026 if the selected link is continuous or stepped. Information on the Mixed sequence advance mode is given later. "0" flags continuous, "1" flags stepped.

Figure 5-7 shows an example of a 4-step sequence of which the first waveform is made of segment 1, which will loop 20 times; segment 2, looping 500 times; segment 3 looping 25 times and segment 4, which

will loop 20 times. Also notice that mixed mode advance flag has been assigned to segment 4. This flag will cause the sequence to stop on segment 4 until the 8026 will be triggered for the next sequence cycle.

There are control buttons on the right of the sequence table. The **Channel#** field lets you select the active channel. If "1" is displayed, then the table you'll be constructing will affect channel 1. Likewise, select "2" if you want to program the table for channel 2.

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.

The **Clear All** key will remove all steps from the table and will let you start designing your sequence table from scratch.

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 **Store** 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.

The Trigger Panel

The Trigger panel, as shown in Figure 5-8, is invoked either by pressing the TRIG button on the Panels bar, or clicking on one of the Triggered, Gated, or Burst buttons on the Main Panel. Note that if you invoke the Trigger Panel from the Panels menu, the 8026 will not change its operating mode. On the other hand, if you select the Trigger option from the Main Panel, besides invoking the Trigger Panel, the 8026 will immediately change its operating mode to triggered. The functional groups in the Trigger Panel are described below.

Trigger Parameters

The Trigger Parameters group contains two parameters for each channel: Phase and Count. To access the required parameter, click on the button below the Channel1 or Channel 2 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, press Execute to update the 8026 with the new reading.

Note

Normal color of the digital reading is dark blue. If you modify the reading, the color changes to a lighter shade of blue, indicating that the 8026 has not been updated yet

with the new parameter. Pressing Execute will update the instrument and will restore the color of the digital readout to dark blue, indicating that the displayed value is the same as the generator setting.

Also note that the digital readout has an autodetect mechanism for the high and low limits. You cannot exceed the limits if you are using the dial but only if you use the keypad. In case you do, the program will not let you download an illegal parameter and you'll be requested to correct your setting.

Slope   The Slope group lets you select edge sensitivity for the trigger input of the 8026. If you click on Positive, the instrument will trigger on the rising edge of the trigger signal. Likewise, if you click on Negative, the instrument will trigger on the falling edge of the trigger signal.

Source   The Source group Provides access to trigger source parameters. Click on External to enable the rear-panel TRIG IN connector. If you select the external option, you must connect your stimulating signal to the trigger input. You can also use the Manual button to trigger the instrument. If you select the Internal option, you will be replacing the external signal with an internal, free-running, synthesizer that is connected internally to the trigger input. The period of the internal synthesizer is programmable using the Timer button in this group.

Arm   The Arm group is really a semi-orphan on the Trigger Panel. This is the only function that actually will work in continuous mode however, since the arm commands cause interruption of the 8026 output and arm control is done with the trigger input, it was placed in the trigger section of ArbExplorer. While modifying Arm commands, bear in mind that the instrument must be in continuous mode only.

The arm function requires definition of breakpoints. Additional information on the arm function is given in Chapter 3 of this manual. To turn ARM on and off, click on the **State** button. Select the Position LED to program **breakpoint** position. The **Positive** on **Negative** slope options will determine edge sensitivity for the trigger input.

There are two additional buttons at the right top corner of the Trigger Panel. You can close the Trigger Panel by clicking on the Close key, or return to the main panel and leave the Trigger Panel in the background if you click on the "To Main Panel" key.

The Modulation Panel

The Modulation panel, as shown in Figure 5-8, is invoked either by pressing the MOD button or "To Modulation Panel" button on the Main Panel. The modulation panel provides access and control for a number of functions: FM (frequency modulation), FSK (frequency shift keying) and Sweep. The 8026 can perform only one of these functions at a time and ArbExplorer will generate an error if you try to change the state to on for more than one modulation option.

The functional groups in the Modulation Panel are divided to three parts: Frequency Modulation, FSK and Sweep. These groups are described below.
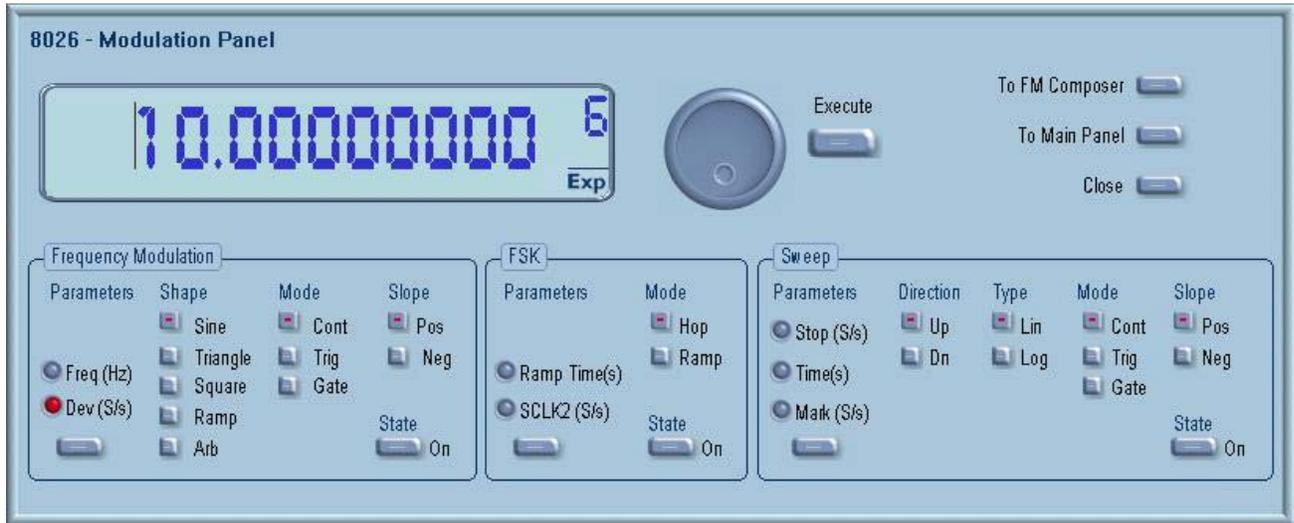
Figure 5-8. The Modulation Panel

Frequency Modulation   The Frequency Modulation group contains parameters for controlling the sample clock. To turn the FM function on and off, click on the State button in this group. There are sub-groups in the FM block: Parameters, Shape, Mode and Slope.

Shape – Determines how the frequency will change. There are four standard patterns that are built into the instrument: sine, triangle, square and ramp, which could be selected from a built-in lookup table. The arbitrary waveform shape must be loaded to the instrument from an external application, or from the FM Composer. Information on the standard and arbitrary FM functions is given in Chapter 3 of this manual. Click on the button next to the required modulating waveform shape to select it.

Parameters – Allow adjustment of waveform parameters that are selected from the Shape sub-group. Notice that the parameters are changing as you select different shapes. 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, press Execute to

update the 8026 with the new reading.

Note

Normal color of the digital reading is dark blue. If you modify the reading, the color changes to a lighter shade of blue, indicating that the 8026 has not been updated yet with the new parameter. Pressing Execute will update the instrument and will restore the color of the digital readout to dark blue, indicating that the displayed value is the same as the generator setting.

Also note that the digital readout has an autodetect mechanism for the high and low limits. You cannot exceed the limits if you are using the dial but only if you use the keypad. In case you do, the program will not let you download an illegal parameter and you'll be requested to correct your setting.

Mode- The Mode sub-group let you select one of the FM operating modes. You have a choice of continuous, triggered and gated FM. Unlike the main trigger function, the trigger modes for the FM function operate in continuous mode. For example, if you place the instrument in triggered FM, the instrument will idle at the carrier frequency and execute one FM cycle when triggered. Refer to the appropriate section in this manual to learn more about triggered FM.

Slope – The Slope sub-group let you select edge sensitivity for the trigger input of the 8026. If you click on Positive, the instrument will trigger on the rising edge of the trigger signal. Likewise, if you click on Negative, the instrument will trigger on the falling edge of the trigger signal.

FSK The FSK group contains parameters for controlling the FSK function. To turn the FSK function on and off, click on the State button in this group. There are two other sub-groups in the FSK block: Parameters, and Mode.

Mode – The Mode group lets you select between two shift options: hop and ramped. In hop mode, the sample clock hops to the shift frequency without delays. In ramped mode, the sample clock ramps to the shift frequency at a rate determined by the Ramp Time parameter. Click on the option you prefer to select between hop and ramped modes.

Parameters – Allow adjustment of Ramp Time and Shifted Sample Clock frequency parameters. 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, press Execute to update the 8026 with the new reading.

Sweep

The Sweep group contains parameters for controlling sweep options. To turn the Sweep function on and off, click on the State button in this group. There are sub-groups in the Sweep block: Parameters, Direction, Type, Mode and Slope.

Parameters – Allow adjustment of Sweep Stop and Sweep Time. You can also place a marker at a position programmed by the Mark parameter. 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, press Execute to update the 8026 with the new reading.

Note

Normal color of the digital reading is dark blue. If you modify the reading, the color changes to a lighter shade of blue, indicating that the 8026 has not been updated yet with the new parameter. Pressing Execute will update the instrument and will restore the color of the digital readout to dark blue, indicating that the displayed value is the same as the generator setting.

Also note that the digital readout has an autodetect mechanism for the high and low limits. You cannot exceed the limits if you are using the dial but only if you use the keypad. In case you do, the program will not let you download an illegal parameter and you'll be requested to correct your setting.

Direction – Use the keys in this sub-group to program sweep direction. Up select sweep from 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.

Type – Use these keys to select sweep step from two options: linear and logarithmic.

Mode- The Mode sub-group let you select one of the Sweep operating modes. You have a choice of continuous, triggered and gated sweep. Unlike the main trigger function, the trigger modes for the sweep function operate in continuous mode. For example, if you place the instrument in triggered sweep, the instrument will idle at the carrier frequency and execute one sweep cycle when triggered. Refer to the appropriate section in this manual to learn more about triggered

sweep.

Slope – The Slope sub-group let you select edge sensitivity for the trigger input of the 8026. If you click on Positive, the instrument will trigger on the rising edge of the trigger signal. Likewise, if you click on Negative, the instrument will trigger on the falling edge of the trigger signal.

There are three additional buttons at the right top corner of the Modulation Panel. You can close the Modulation Panel by clicking on the Close key, or return to the main panel and leave the Modulation Panel in the background if you click on the "To Main Panel" key. The "To FM Composer" button will invoke the FM composer program that will enable you to create and edit user-defined waveforms that will be used as modulating waveforms. The FM Composer program is described in greater details later in this chapter.

The Utility Panel

The Utility panel, as shown in Figure 5-9, is invoked either by pressing the UTIL button or "To Utility Panel" button on the Main Panel. The Utility panel provides access to general instrument controls that are not directly related to waveform generation. Through this panel you can control multiple instruments and multiple channels. If you made programming errors, you can use this panel to clear the error queue, or reset the 8026 all together. The functional groups in the Utility are described below.

System Commands

Commands in this sub-group will let you **Reset** the entire instrument to factory defaults, **Clear Error Queue**, if you made programming errors and want to remove the error list from the queue. You can also use the **Manual Trigger** key to trigger the 8026 in case you do not have an external trigger stimuli connected to the TRIG IN connector.

Filter

The Filter group lets you place filters at the output path. There are two sets of filters – one for each channel. Click on 25 MHz, 50 MHz or All to select both. Click on Off to remove disable all filters.

Figure 5-9. The Utility Panel

Inter-Channel Control   The Inter-Channel Control group lets you place an offset between channel 2 and 1. The offset is applied between the leading edges of the two channels and is given in units of waveform points. To adjust the Offset parameter, click on the button below parameter. The LED next to the Offset turns on. The offset value 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, press Execute to update the 8026 with the new reading.

Note

Normal color of the digital reading is dark blue. If you modify the reading, the color changes to a lighter shade of blue, indicating that the 8026 has not been updated yet with the new parameter. Pressing Execute will update the instrument and will restore the color of the digital readout to dark blue, indicating that the displayed value is the same as the generator setting.

Also note that the digital readout has an autodetect mechanism for the high and low limits. You cannot exceed the limits if you are using the dial but only if you use the keypad. In case you do, the program will not let you download an illegal parameter and you'll be requested to correct your setting.

Multi-Instruments Sync Control

The Multi-Instrument Sync Control group lets you synchronize between multiple instruments and program phase offset between instruments. The phase offset is applied between the leading edges of the two instruments and is given in units of waveform points. To adjust the Ph. Offset parameter, click on the button below parameter. The LED next to the Ph. Offset turns on. The offset value 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, press Execute to update the 8026 with the new reading.

Mode – The Mode buttons let you select between **Master** and **Slave** modes. Special cables are required to connect multiple instruments in a daisy-chain link and the Master/Slave options will work only when the instruments are connected properly to each other. Use the information in Chapter 3 to find out more about multi-instruments synchronization.

State – The State button will turn the couple mode on and off.

There are two additional buttons at the right top corner of the Utility Panel. You can close the Utility Panel by clicking on the Close key, or return to the main panel and leave the Utility Panel in the background if you click on the "To Main Panel" key.

The System and Command Editor

ArbExplorer requires certain conditions to work well with the 8026. For example, The computer you are using must be connected to the instrument through a known and well-defined communication link, such as, GPIB or RS232. Not only that the instrument must be connected properly but also, software and registers must be set properly for the generator to function properly. It is also possible that your computer is running smoothly but the 8026 has problems communicating with your computer. The System and Command Editor dialog box has all the tools needed to make sure that the 8026 is connected properly and that it responds correctly to machine commands.

Invoke the System and Command Editor from the System command at the top of the screen. Dialog box, as shown in Figure 5-10 will pop up. There are three main functions that you can execute using this dialog box: Set your communication link, Reset and test your communication with the 8026 and, last but no least, send SCPI commands to the instrument and see an immediate response. Description of the various elements of this dialog box is given below.
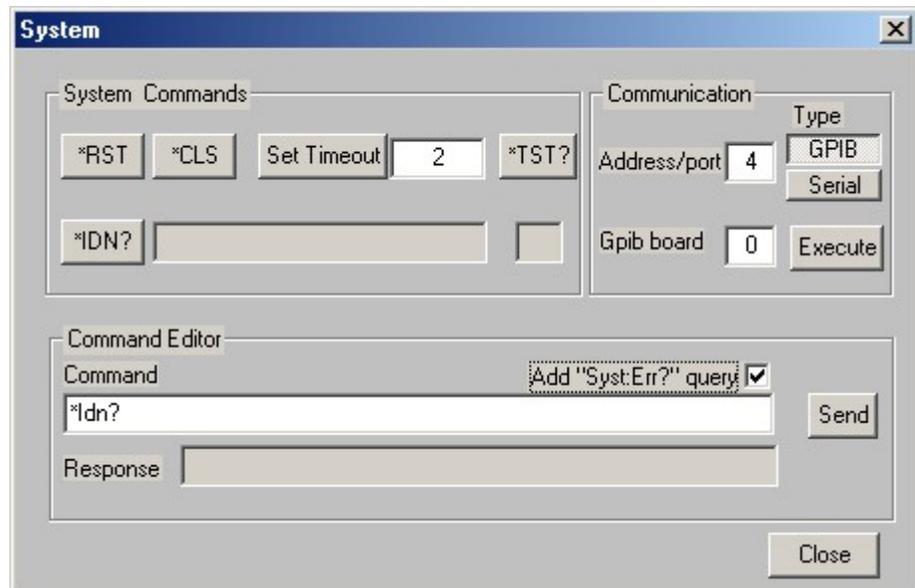
Figure 5-10. System Control and the Command Editor

Communication   The Communication group lets you select the proper communication link for connecting the 8026 to the computer. In general, ArbExplorer does that automatically for you if you use the Refresh Communication command. However, in cases where you operate with both RS232 and GPIB and want to test both options, you can do this by clicking on the appropriate button that selects the communication type. Remember to click on Execute after you change link type to update the program with your new selection.

If you are using GPIB, use the Address/Port field to change your GPIB address. The 8026 ships with address 4 and so is the default address in this program. You may change the address as long as you stay within the address limits (1 through 31) but make sure that you update your 8026 to the same address as you select in this field. This field is also useful in cases where you have more than one instrument connected to your GPIB bus. Just change the address and click on the execute button for this address to become active.

Most computers have two RS232 ports, designated as COM1 and COM2. For RS232 communication, select 1 or 2 to place in this field but make sure that your instrument is connected properly to the select serial port. Note that ArbExplorer does not limit you to certain port numbers, so if your computer can handle multiple ports, just make sure that you have the correct port identification set in this field.

System Commands   The System Commands group lets you use common commands and test the 8026 for proper operation. The Time Out field sets system timeout in units of seconds. The time out is required by the GPIB ink in case your instrument does not communicate properly with on the bus. To avoid bus lockout, set the time out to small values. The default time out setting is 2. The ***RST**, ***CLS** buttons are used for clearing error queue, or resetting the instrument to factory defaults.

The ***TST?** Invokes the 8026 selftest routine. You should use this test when you suspect that something is wrong with your instrument. The 8026 executes this test and generates a response that is placed in the field below the *TST? button. The response to the selftest command is numeral. Any response, besides "0", means that your instrument failed the selftest. Additional information on the *TST? command is available in chapter 4 of this manual.

WARNING

The *TST? Invokes hardware selftest routine. This test is destructive to the last setting of the 8026. Use this test routine only when absolutely necessary to ascertain functional failure of the 8026.

The *IDN? queries the generator's identity. The response is placed in the field next to the *IDN? button. The response is organized into four fields, separated by commas, as shown below:

```
Tabor Electronics,8026,0,1.0
```

Command Editor   Low-level SCPI commands and queries can be directly sent to the 8026 from the **Command** field and the 8026 will response to queries in the **Response** field. The command editor is very useful while developing your own application. Build your confidence or test various commands using the command editor. This way you can assure that commands or syntax that you use in your application will respond exactly the same way as it responds to the editor commands. A complete list of SCPI commands is available in Chapter 4 of this manual.

The Waveform Composer

Being an arbitrary waveform generator, the 8026 has to be loaded with waveform data before it can start generating waveforms. The waveform generation and editing utility is part of ArbExplorer and is called – The Waveform Composer. This program gives you tools to create definitions for arbitrary waveforms. It can also convert coordinates from other products, such as, oscilloscopes and use them directly as waveform data. 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 8026 using the Waveform Composer.

You can invoke the Waveform Composer program from two places: On the Panels bar click on WAVE, or from the Arbitrary & Sequence Panel click on "To Wave Composer". Figure 5-11 shows the waveform composer, immediately after you first invoke this utility.
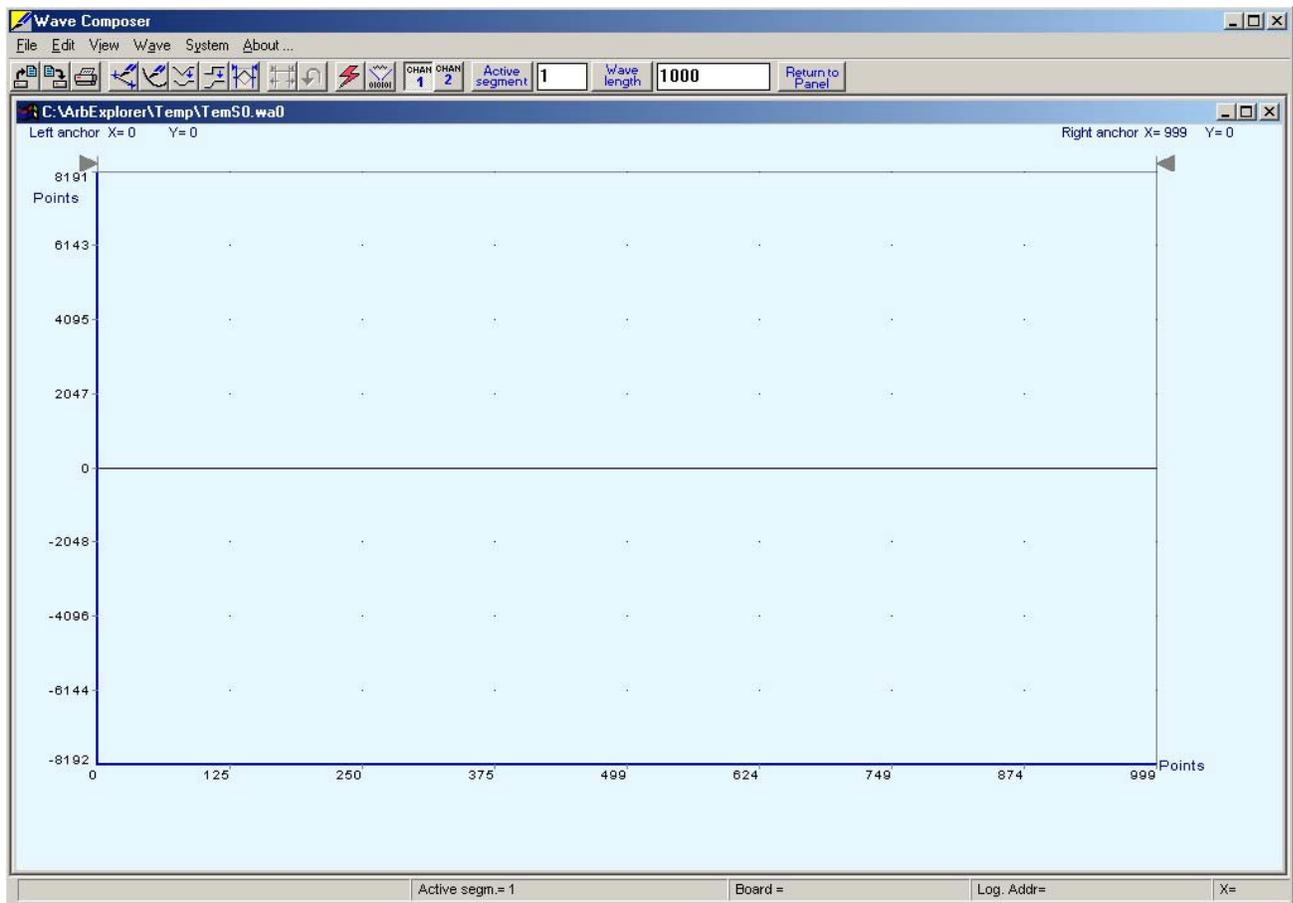

Figure 5-11. The Wave Composer Opening Screen

The Wave Composer has three sections: Commands bar, Toolbar and Waveform screen. Refer to Figure 5-11 throughout the description of these parts.

The Commands bar

The commands bar provides access to standard Windows commands such as File and View. In addition, there are ArbExplorer-specific commands such as Edit, Wave and System.

In general, clicking on one of the commands, opens a dialog box with an additional list of commands. Then, clicking on an additional command, may open a dialog box, or generate an immediate action. For example, Clicking on File and then Exit will cause an 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 Commands bar are listed and described below.

File Commands

The File command has 4 command lines that control waveform files. Also use this command to print the active waveform, or exit the wave 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 the New Waveform 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… command will let you brows your disk for previously saved waveform files and load these waveforms to the waveform area. This command is also very useful for converting waveform files to format that is acceptable by the Wave Composer. The Open Waveform command can convert ASCII. *CSV (comma delimited text), *PRN (space delimited text) and *.0* (Tabor binary format). For additional information on files format, contact your nearest Tabor representative.

Save Waveform (Binary)

The Save Waveform command will store your active waveform in your 8026 directory, as a binary file with an *.wav extension. If this is the first time you save your waveform, the Save Waveform As… command will be invoked automatically, letting you select name, location and format for your waveform file.

Save Waveform As…

Use the Save Waveform As… command the first time you save your waveform. It will let you select name, location and format for your waveform file. The Save Wave As… dialog box is shown in Figure 5-

12.
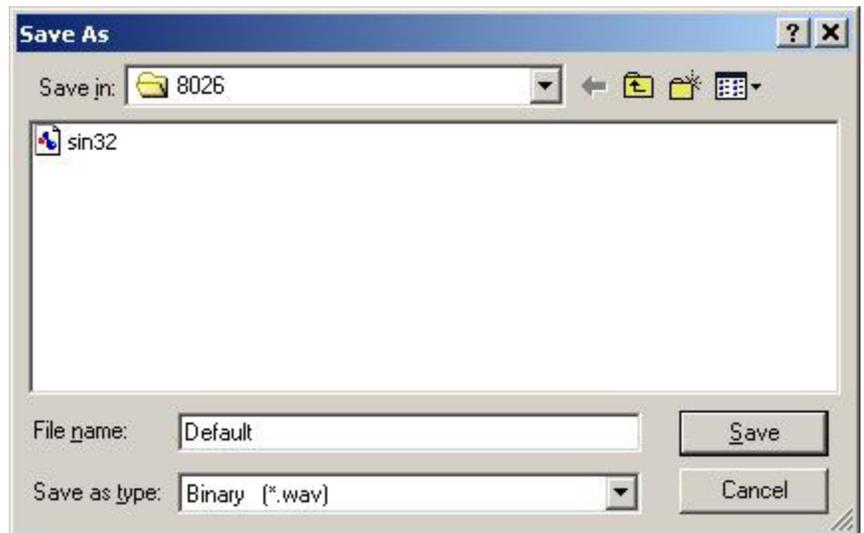
Figure 5-12. The Save Wave AS… Dialog Box

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 Wave Composer session and takes you back to the Panels screen. If you made changes to your waveform since it was last saved, the Wave Composer will prompt you to Save or Abandon changes these changes.

Edit Commands   The Edit commands are used for manipulating the waveform that is drawn on the screen. The editing commands are explained in the following paragraphs.

Autoline

The Autoline command lets you draw straight-line segments. To draw a line using this command click the left mouse button at the start point. Click again at the next point and then click on the right mouse button to terminate this operation.

Sketch

The Sketch command lets you draw free-hand segments. To draw a line using this command click and hold the left mouse button at the start point. Release the mouse button when you want to stop and then click on the right mouse button to terminate this operation.

Smooth

The Smooth command lets you smooth out rough transitions on your waveform. This is done mathematically by multiplying waveform coordinates by the non-linear portion of a cubic parabola.

The Smooth operation is done on segments of the waveform that are bound by anchors. Anchor operation is described later in this chapter. Place you anchors on the left and right of your waveform segment and select the Smoot command. The waveform will change its shape immediately to follow the mathematical pattern of a parabolic curve.

Note that small segments with fast transitions, when combined with parabolic expressions have tendencies to generate even larger transitions. Therefore, make sure you omit such sections of the waveform when you use this operation.

Filter

The Filter used with this command is moving average. This is done by recalculating each point as an average of symmetrical number of adjacent points. When you select the Filter command, a dialog box pops up, letting you program the filter spacing in number of adjacent points. You can filter the entire waveform, or you may chose to filter a segment of the waveform by placing the anchors as boundaries on the left and right of the segment.

Invert

The Invert command lets you invert the entire waveforms, or marked segments of waveforms. The waveform is inverted about the 0-point axis.

Unmark

The unmark command removes the anchors from the waveform screen and resets anchor positions to point 0 and the last waveform point.

Undo

The Undo command undoes the last editing operation.

View Commands   The View commands have commands that let you view various options on the waveform area. The View commands include: View All, Channel 1 Waveform, Channel 2 Waveform, Return to Panel and Hide/Show Toolbar. Description of the view commands is given in the following.

View All

The View All command is actually a zoom out action that restores the waveform screen to display the complete waveform. You may zoom in on portions of the waveform by clicking and holding the mouse at one corner and releasing the mouse button at the other corner. Figure 5-13 shows a zoomed-in portion of the waveform with the Waveform

Map that is automatically invoked when you zoom in.

Looking at the Waveform Map, you'll see that the white portion is the zoomed area. Click and hold on the white area and move your cursor around and the waveform screen will be updated accordingly.

While zoomed in you can perform autoline and sketch editing, or zoom-in further by clicking and holding the mouse at one corner and releasing the mouse button at the other corner.
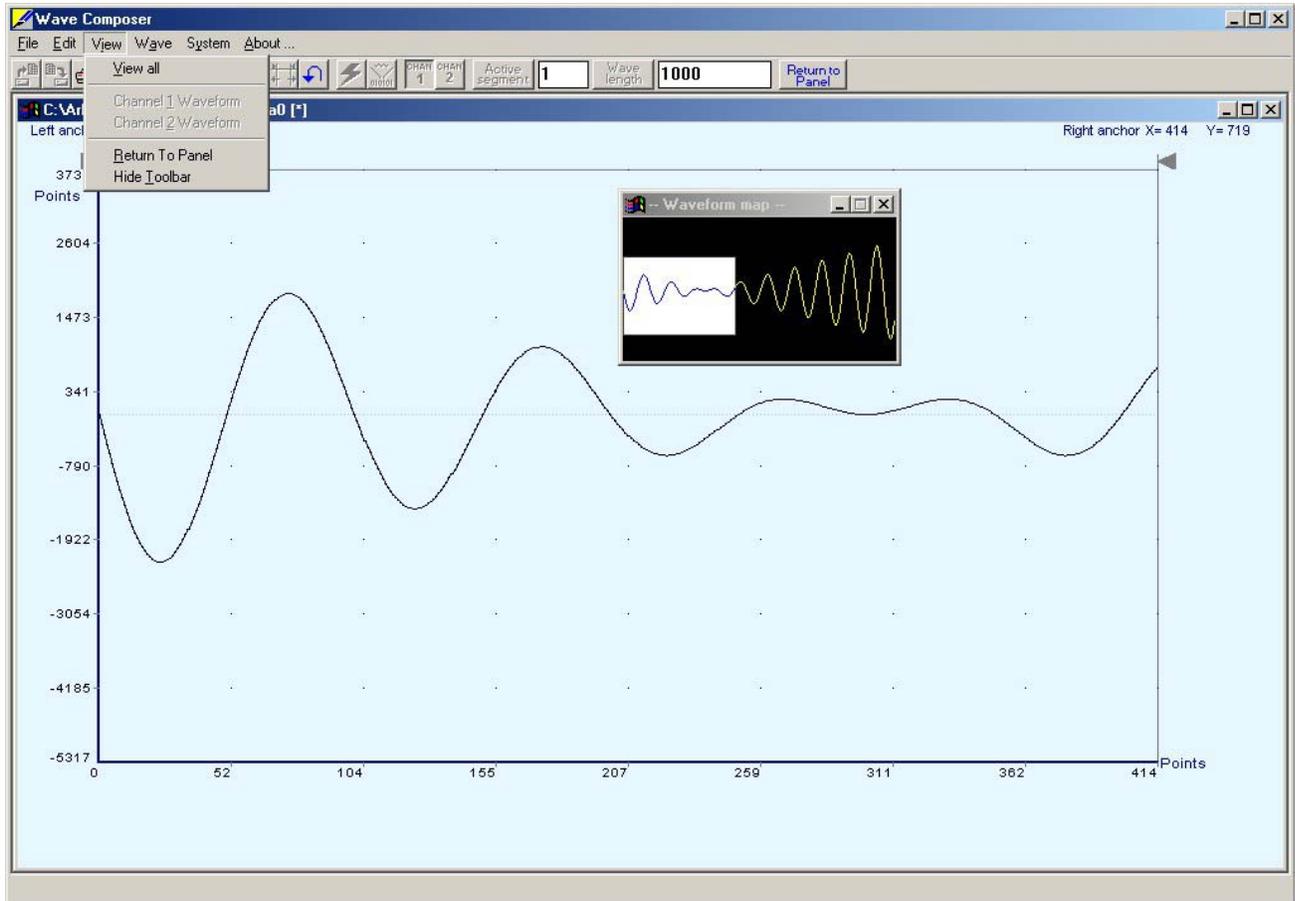

Figure 5-13. Zooming In on Waveforms

Channel 1 Waveform

The Channel 1 Waveform command updates the waveform screen with the Channel 1 waveform. If you have not yet generated a waveform for channel 1, the waveform screen will show a dc level at vertical point 0.

Channel 2 Waveform

The Channel 2 Waveform command updates the waveform screen

with the Channel 2 waveform. If you have not yet generated a waveform for channel 2, the waveform screen will show a dc level at vertical point 0.

Return to Panel
The Return to Panel command will close the wave composer and will return control to the panel screen.

Hide Toolbar
The Hide Toolbar toggles between Hide and Show.

Wave Commands The Wave commands let you create waveforms on the screen. The Wave command has a library of 8 waveforms: Sine, Sawtooth, Square, Sinc, Gaussian, Exponent, Pulse, and Noise. It also lets you create waveforms using the Equation Editor. Information how to create waveforms using the Wave commands is given below.

Creating Waveforms From the Built-in Library
You can create any waveform from the built-in library using the Wave command. Clicking on one of the Wave options will open a dialog box. An example of the Sine waveform dialog box is shown in Figure 5-14. 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 as shown in Figure 5-14 will appear. 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 – 14-bit of vertical define 16,384 incremental steps. The Amplitude parameter defines how many of these steps are used for generating the sine. The example is showing sine waveform with maximum peak-to-peak amplitude. Any number below the maximum will generate an attenuated sine.

Start Phase – The start phase parameter defines the angle of which

the sine will start. The example shows 90° start phase.

Power – The example shows sine cubed. Sine to the power of 1 will generate a perfect sine. Power range is from 1 through 9.
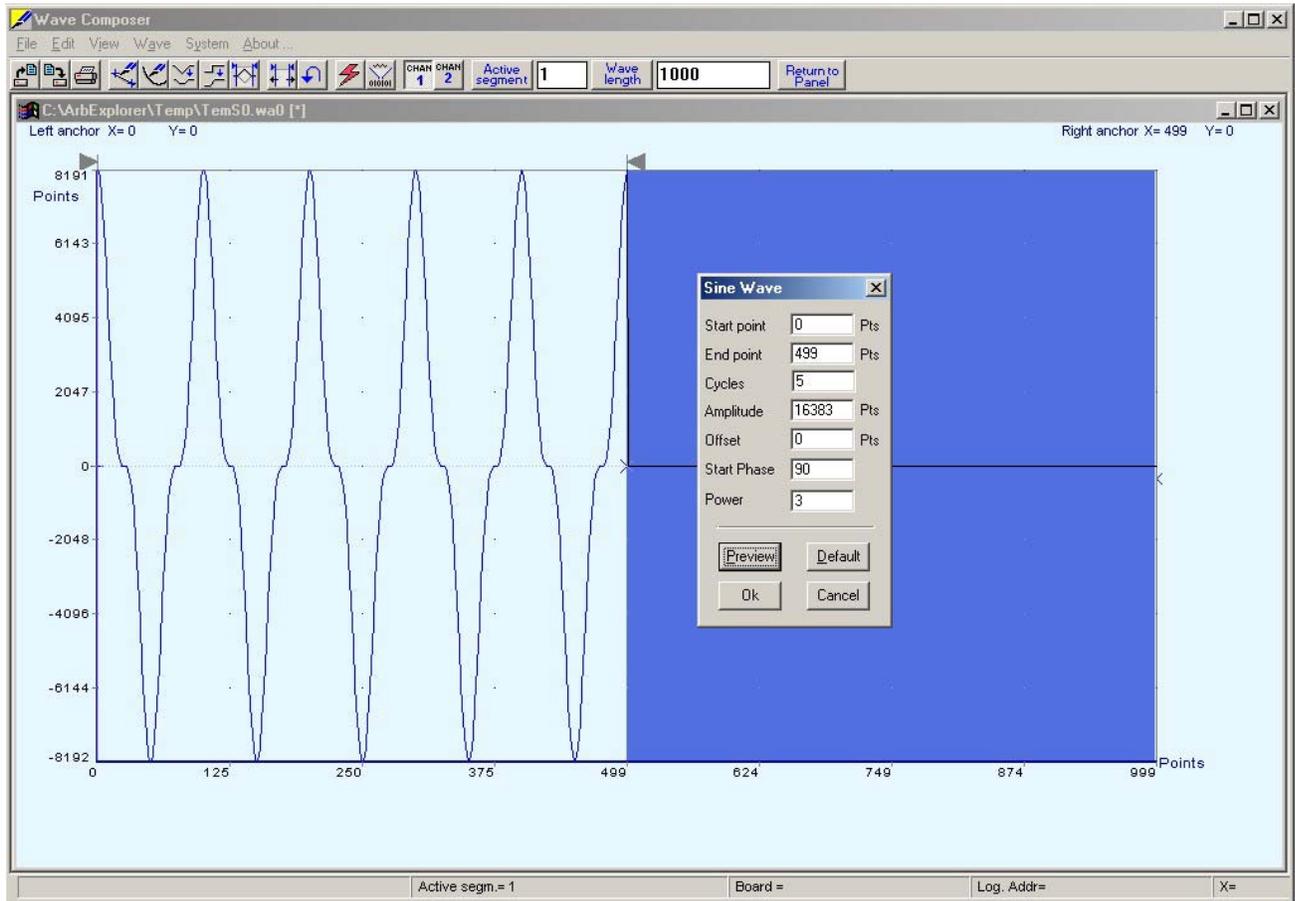

Figure 5-14. An Example of Generating Sine Waveforms From the Built-in Library.

System Commands The System commands let you do two actions: Delete all segments and download the active waveform to the selected channel. Both of these commands are also available as icons on the toolbar.

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 5-15 and the individual icons are described in the following.

Figure 5-15. The Toolbar Icons

This icon represents the Open Waveform file command. When you click on this icon, the Open Waveform dialog box will open and will let you select a waveform file to display on the waveform screen.

This icon will save the file you currently have on your waveform screen. The file will be saves in *.wav format that is unique to the wave composer.

Use this icon to print your waveform screen on your default printer. 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.

Click on this icon to start the autoline operation. You can exit the autoline mode by clicking on this icon again, or by clicking on the right mouse button. The Autoline command lets you draw straight-line segments. To draw a line using this command click the left mouse button at the start point. Click again at the next point and then click on the right mouse button to terminate this operation.

The sketch mode is invoked using this icon. Click on this icon again, or click your right mouse button to terminate the sketch mode. The Sketch command lets you draw free-hand segments. To draw a line using this command click and hold the left mouse button at the start point. Release the mouse button when you want to stop and then click on the right mouse button to terminate this operation.

Clicking on this icon will invoke the smooth function. The Smooth command lets you smooth out rough transitions on your waveform. This is done mathematically by multiplying waveform coordinates by

the non-linear portion of a cubic parabola.

The Smooth operation is done on segments of the waveform that are bound by anchors. Anchor operation is described later in this chapter. Place you anchors on the left and right of your waveform segment and select the Smoot command. The waveform will change its shape immediately to follow the mathematical pattern of a parabolic curve.

Note that small segments with fast transitions, when combined with parabolic expressions have tendencies to generate even larger transitions. Therefore, make sure you omit such sections of the waveform when you use this operation.

This icon will filter operation. The Filter used with this command is moving average. This is done by recalculating each point as an average of symmetrical number of adjacent points. When you select the Filter command, a dialog box pops up, letting you program the filter spacing in number of adjacent points. You can filter the entire waveform, or you may chose to filter a segment of the waveform by placing the anchors as boundaries on the left and right of the segment.

An easy way of inverting your waveform is by using the invert function. The Invert command lets you invert the entire waveforms, or marked segments of waveforms. The waveform is inverted about the 0-point axis.

Click on this icon to removes the anchors from the waveform screen and resets anchor positions to point 0 and the last waveform point.

Use this icon to undo your last editing operation. Only one step will back up.

Use extreme caution when using this icon because it will delete all of your segments from your waveform memory.

When you click on this button, the waveform that is displayed on your waveform screen will be downloaded to the active segment on the selected channel.

CHAN 1 and CHAN 2 are used for selecting the active channel. The 8026 has two separate channels that accept individual waveforms. By selecting one of these buttons you prepare the wave composer for the next download command. If the CHAN 1 button is depressed, then the waveform will be downloaded to active segment on channel 1.

Active segment 1 Use this edit field to define the active segment you

intend to program. The 8026 lets you program up to 4096 segments, of which each could be programmed with a unique waveform length. To select a new active segment click on the edit field, use your backspace or delete keys to clear the field and type in the new number. To complete the selection, click on the Active Segment button.
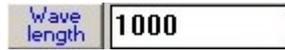
Wave length | **1000** | The wavelength field lets you program the length of the active segment. You can program up to 4096 individual segments that each has a unique length. Minimum segment length is 16 points and maximum is 4,194,303 ("0" is counted as a point for a total of 4,194,304). To program a new length for the active segment click on the edit field, use your backspace or delete keys to clear the field and type in the new number. To complete the selection, click on the Wavelength button.

Return to Panel The last icon on the Toolbar is the Return to Panel button. When you click on this button, the program will move the wave composer to the background and panels to the front.

The Waveform Screen

Waveforms are created and edited on the waveform screen. Figure 5-16 shows an example of a waveform created using the equation editor and the anchors to limit generation of the waveform between points 100 and 900. The various elements of the waveform screen are described below.

The waveform screen has two axes – vertical and horizontal. Both axes are divided into points.

The vertical axis is labeled from –8192 through 8191 for a total of 16,384 point. This number represents 14 bits of vertical resolution and cannot be changed because it is critical to the range of which the 8026 operates.

The horizontal axis, by default has 1000 points (from point 0 to 999). This number can be changed using the Wave Length field in the Toolbar. The maximum length depends on the option installed in your instrument. The wave composer will let you define the horizontal axis to a maximum of 4,194,304 (4 Meg words).
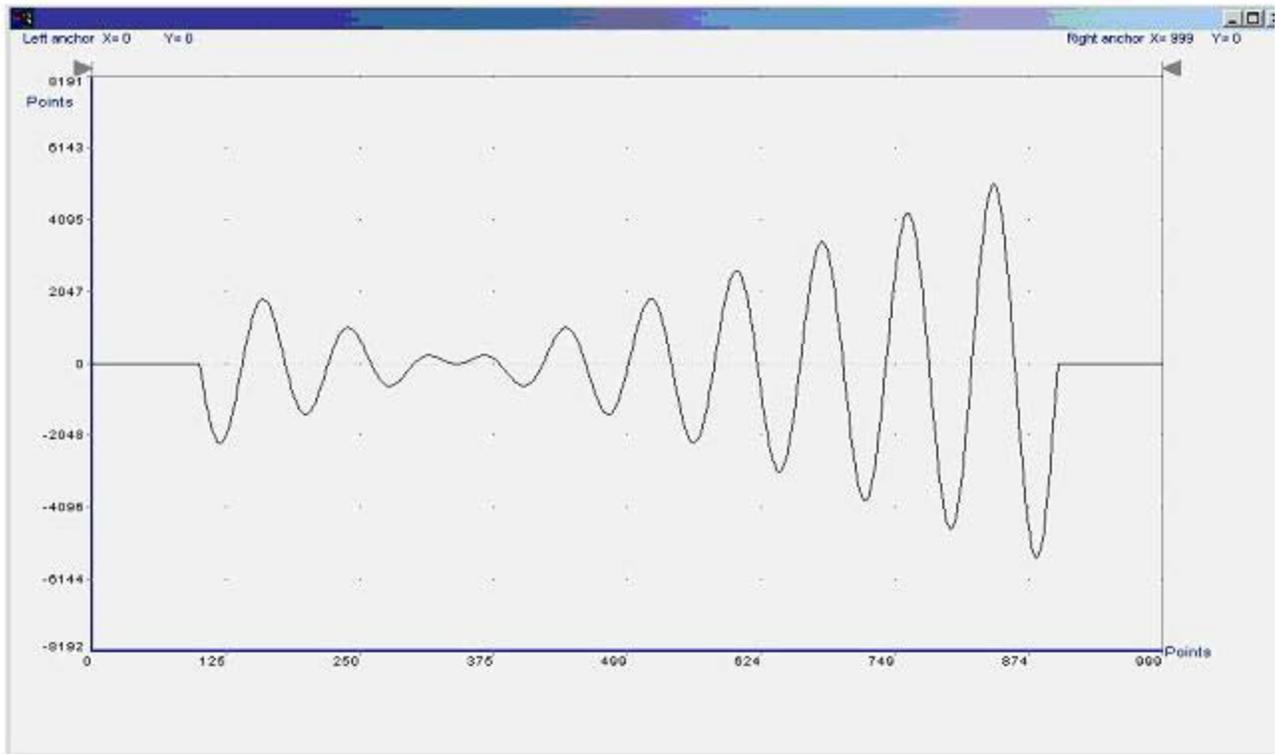
Figure 5-16. The Waveform Screen

Notice on the left top and on the 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 where your waveform will be created. 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) Click and hold your mouse cursor on the left anchor triangle and then drag the curtain to the left position. Do the same for the right anchor. Notice the X and Y coordinates at the top of the waveform screen and how they change to correspond to your anchor placement.

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 5-14.

Finally, when you are done creating and editing your waveform, you can save your work to a directory of your choice. The name at the title

will show you the name you selected for storing your waveform and its path.

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 5-17, you'll see that the vertical axis is marked with sample clock frequencies. You'll see later that as you draw waveforms on the FM composer screen, these waveforms will 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 8026 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 8026 using the FM Composer.

You can invoke the FM Composer from two places: On the Panels bar click on FM, or from the Modulation Panel click on "To FM Composer". Figure 5-17 shows the waveform composer, immediately after you first invoke this utility.

The Wave Composer has three sections: Commands bar, Toolbar and Waveform screen. Refer to Figure 5-17 throughout the description of these parts.

The Commands bar

The commands bar provides access to standard Windows commands such as File and View.

In general, clicking on one of the commands opens a dialog box with an additional list of commands. Then, clicking on an additional command, may open a dialog box, or generate an immediate action. For example, Clicking on File and then Exit will cause an immediate termination of the FM Composer. On the other hand, clicking on Wave and then on Square, will open a Square Wave dialog box that lets you program and edit square wave parameters. The various commands in the Commands bar are listed and described below.
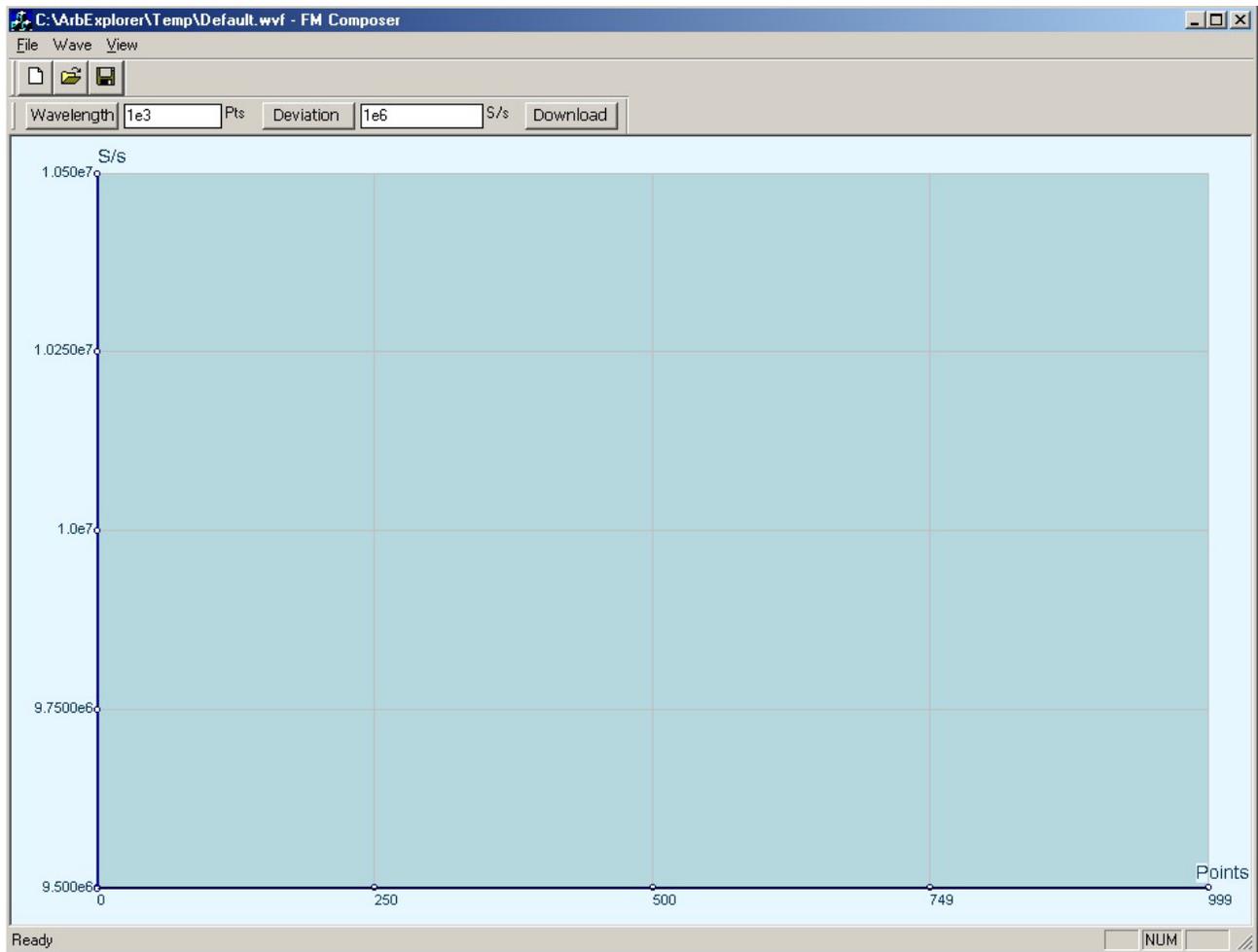
Figure 5-17. The FM Composer Opening Screen

File Commands   The File command has 4 command lines that control waveform files. Also use this command 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… command will let you brows your disk for previously saved waveform files and load these waveforms to the waveform area. This command is also very useful for converting waveform files to format that is acceptable by the Wave Composer. The Open Waveform command can convert ASCII. *ASF (space delimited text). For additional information on files format, contact your

Save Waveform

The Save Waveform command will store your active waveform in your 8026 directory, as a binary file with a *.wvf extension. If this is the first time you save your waveform, the Save Waveform As… command will be invoked automatically, letting you select name, location and format for your waveform file.

Save Waveform As…

Use the Save Waveform As… command the first time you save your waveform. It will let you select name, location and format for your waveform file. The Save As… dialog box is shown in Figure 5-18.
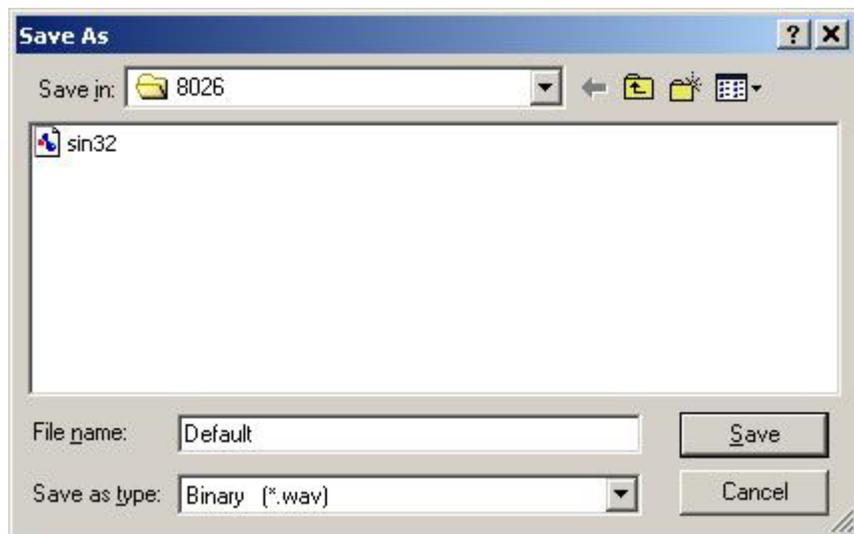
Figure 5-18. The FM Composer Save AS… Dialog Box

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 commands let you create waveforms on the screen. The Wave command 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 commands is given below.

Creating Waveforms From the Built-in Library

You can create any waveform from the built-in library using the Wave command. Clicking on one of the Wave options will open a dialog box. An example of the Sine waveform dialog box is shown in Figure 5-19. 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 as shown in Figure 5-19 will appear. You can now start programming parameters that are available in this box.

Start Point Anchor – 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 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.

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 of 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.
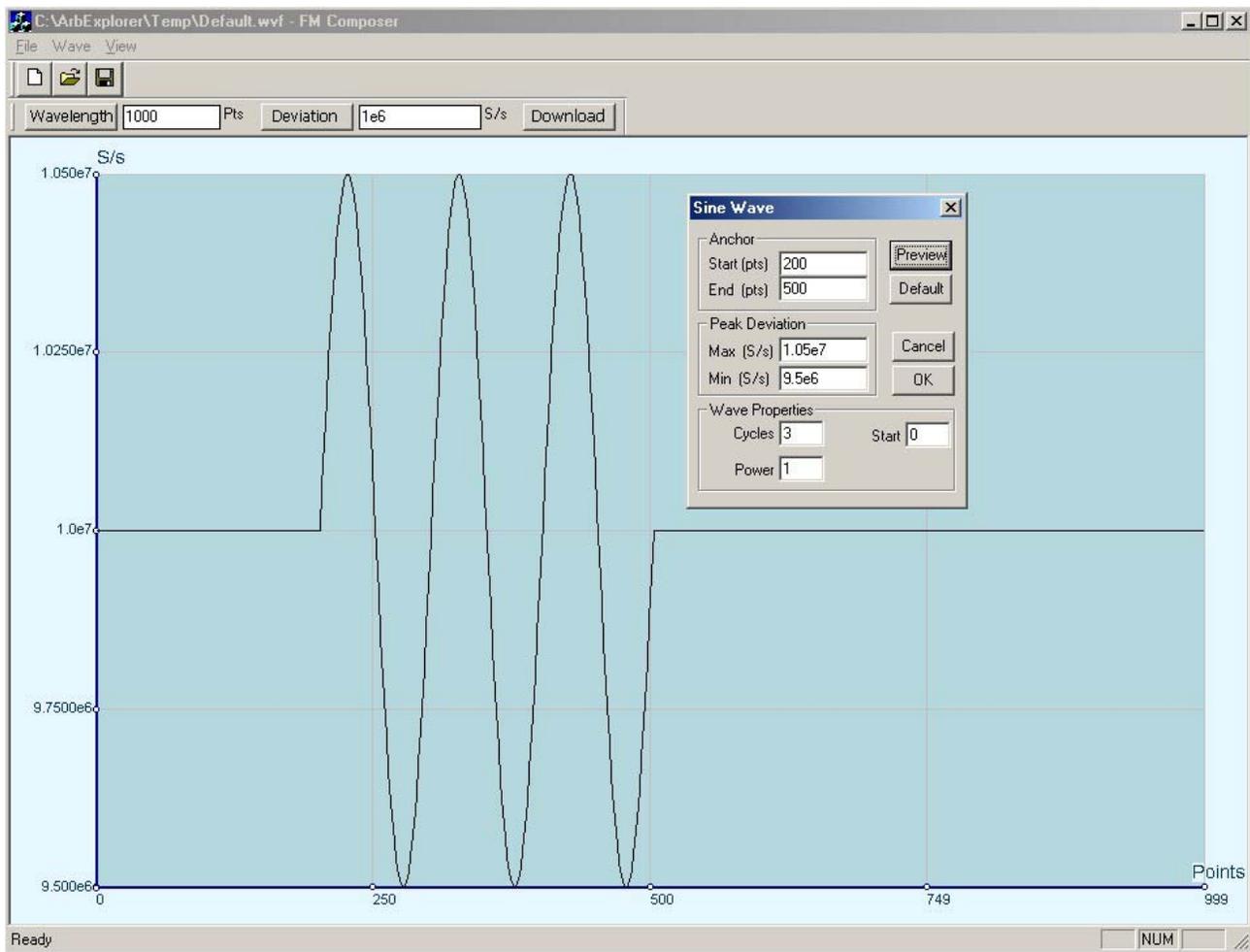
Figure 5-19. An Example of Generating Modulating Sine From the Built-in Library.

Generating Waveforms Using the Equation Editor

One of the most powerful feature within ArbExplorer and probably the feature that will be used most is the Equation Editor. The Equation Editor let you write equations the same way as you would do on a blank piece of paper. The equations are then translated to sequential points that form waveforms and are displayed on the waveform screen. The Equation Editor will detect and inform you on syntax errors and, with its self adjusting feature, will automatically adjust your parameters so that none of the points on your waveform will exceed the maximum scale limits.

When you invoke the Equation Editor, the dialog box, as shown in Figure 5-20 will display. Use the following paragraphs to learn how to use this dialog box and how to write your equations.

There are three main sections in the equation editor plus control buttons. These sections are described below.
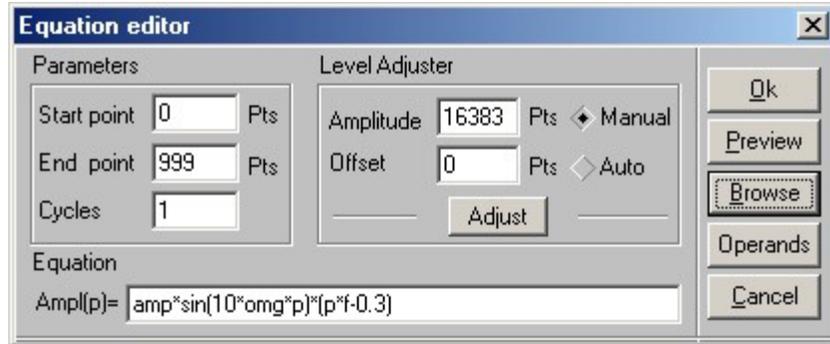
Figure 5-20. The Equation Editor Dialog Box

Parameters   The Parameters section has three fields: Start Point, End Point and Cycles.

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.

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.

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.

For example, look at the equation that is shown in Figure 5-20. This equation will generate a waveform similar to what you can see on Figure 5-16. If you change the offset setting in the Offset field and press the Adjust key, your waveform will offset immediately without changing the equation. The same way, you can also change the amplitude, or both amplitude and offset. If you check the Manual option, you'll have to click on the Adjust button for the Amplitude and Offset 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 and Offset setting.

Equation   You will be using the Equation field for writing your equations. Equation syntax and conventions are discussed in the following.

Control Buttons   There are five 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. The Browse button will open the waveform browser so you can search for pre-stored waveforms to add to your equation. The Operands button will expand the dialog box to show all of the operands that are available for use in the Equation Editor and 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 continuous waveform. However, if you zoom in on a waveform line, you'll see that the points are connected like a staircase. In reality, the 8026 generates its waveforms exactly as shown on the screen however, if the waveform has many horizontal points, the steps are hard to see.

Equations are always computed as a function of the vertical (Amplitude) axis therefore the left side of your equation will always look as Ampl(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

The following paragraphs describe the conventions that are used for writing an equation. To avoid errors, it is extremely important that you make yourself familiar with these conventions before you plan your waveforms.

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-digit precision. For the 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 / per, and numerals from -1E^20 <> 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 summarize the mathematical expressions and their respective abbreviated commands that can be used with the Equation Editor.

Equation Editor Operands

^	Raise to the power
*	Multiply
/	Divide
+	Add
Ä	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 / per
amp	Amplitude in units of points or seconds
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
ln(x)	The natural (base e) logarithm of x
abs(x)	The absolute value of x
-1E^20<>1E^20	Numerals, equation constants

(*)x = argument mathematical expression

After you got yourself familiar with the operands and conventions, you can commence with a few simple equations and see what they do your waveform screen. Once you'll get the feel, you'll be able to explore your own creativity to generate much more complicated and complex waveforms.

Typing Equations

If you remember from your old high school studies, the simplest curve of Y as a function of X is defined by the equation Y=aX+b. You can use the same "technique" to generate straight lines with the Equation Editor. Assuming first that p=0, try this:

Ampl(p)=1000

Press [Preview] and see what you get. Of course, you get an uninteresting line that runs parallel to the X-axis. Now, lets give the line some angle by typing:

Ampl(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: Ampl(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:

Ampl(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:

Ampl(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:

Ampl(p)=6000*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:

Ampl(p)=6000*sin(omg*p*10)

Equation Samples

So far, you have learned how to create two simple waveforms: straight lines and trigonometric functions. Lets see if we can combine these waveforms to something more interesting. Take the straight line equation and add it to the sinewave equation:

Ampl(p)=6000*sin(omg*p*l0)-2*p+2000

Press [Preview]. Your screen should look like Figure 5-21.
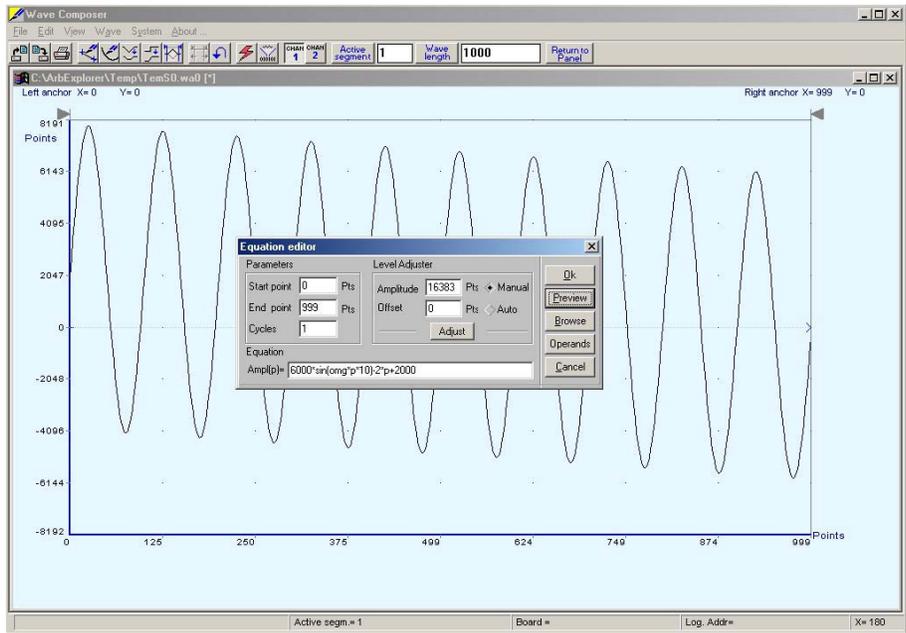
Figure 5-21. Using the Equation Sample to Combine Two Waveforms

Now lets try to modulate two sinewaves with different periods and different start phase. Type this:

Ampl(p)= 8000*sin(omg*p)*cos(omg*p*30)

Press [Preview]. Your screen should look like Figure 5-22.
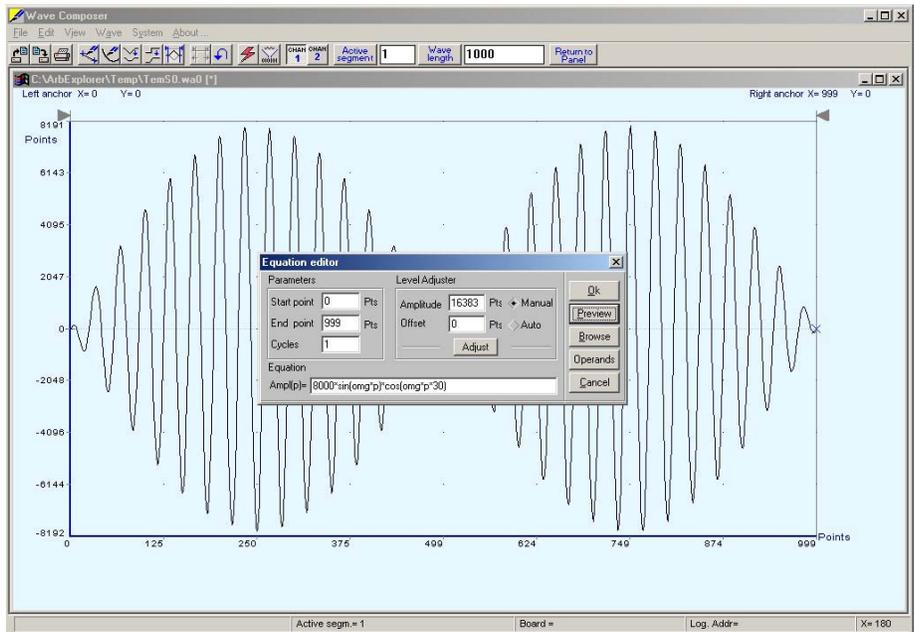
Figure 5-22. Using the Equation Editor to Modulate Sine Waveforms.

In the following example, as shown in Figure 5-23, 20% second harmonic distortion has been added to a standard sinewave. The original waveform had a peak-to-peak value of 5000 points so 20% second harmonic is equivalent to 1000 points. The frequency of the second harmonic is obviously double that of the fundamental, so term +1000*sin(2*omg*t) is added to the original sine wave equation. Use the following equation:

Ampl(p)=5000*sin(omg*t)+1000*sine(2*omg*t)

Press [Preview]. Your screen should look like Figure 5-23.
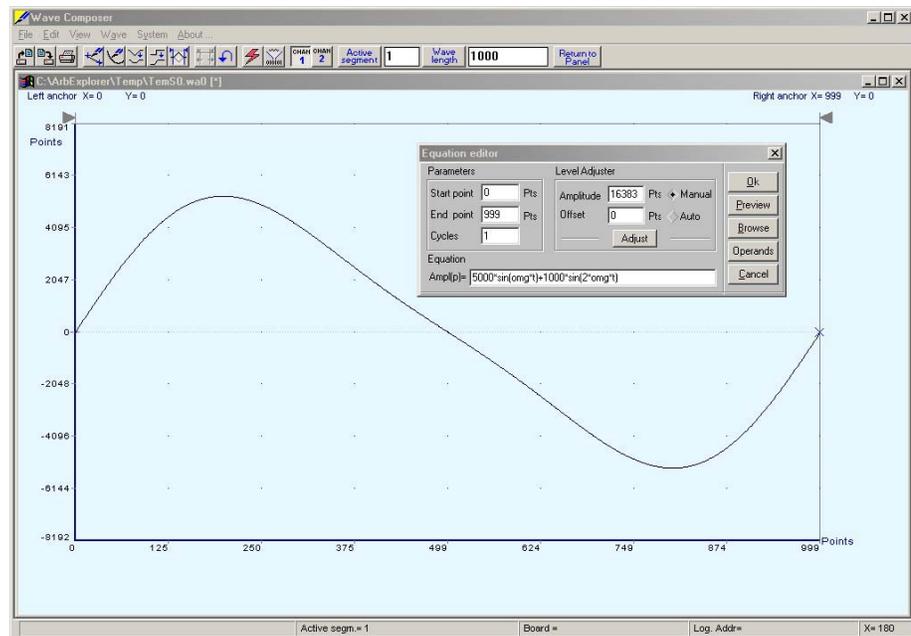
Figure 5-23. Using the Equation Editor to Add Second Harmonic Distortion.

In Figure 5-24 we created 10 cycles of sinewave made to decay exponentially. The original expression for a standard sinewave is multiplied by the term e^(-t/250). Increasing the value of the divisor (200 in this case) will slow down the rate of decay.

Use the following equation:

Ampl(p)=8000*sin(omg*t*10)^e(-t/250)

Press [Preview]. Your screen should look like Figure 5-24.
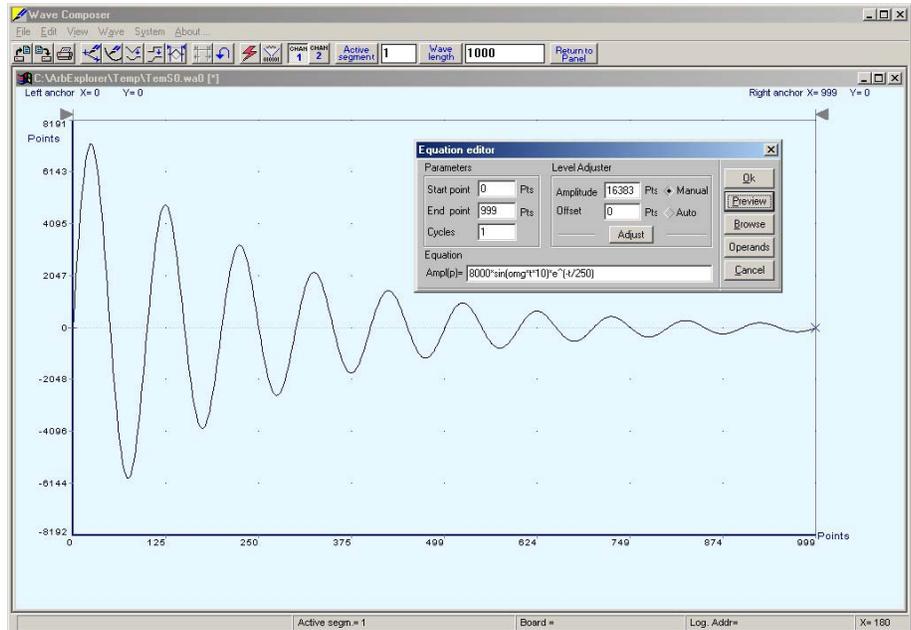
Figure 5-24. Using the Equation Editor to Generate Exponentially Decaying Sinewave

The last example as shown in Figure 5-25 is the most complex to be discussed here. Here, 100 cycles of sinewave 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*t) and the lower sideband by the term 100*cos(90*omg*t).

Use the following equation:

Ampl(p)=5000*sin(100*omg*t)+1000*cos(110*omg*t)-1000*cos(90*omg*t)

Press [Preview]. Your screen should look like Figure 5-25.
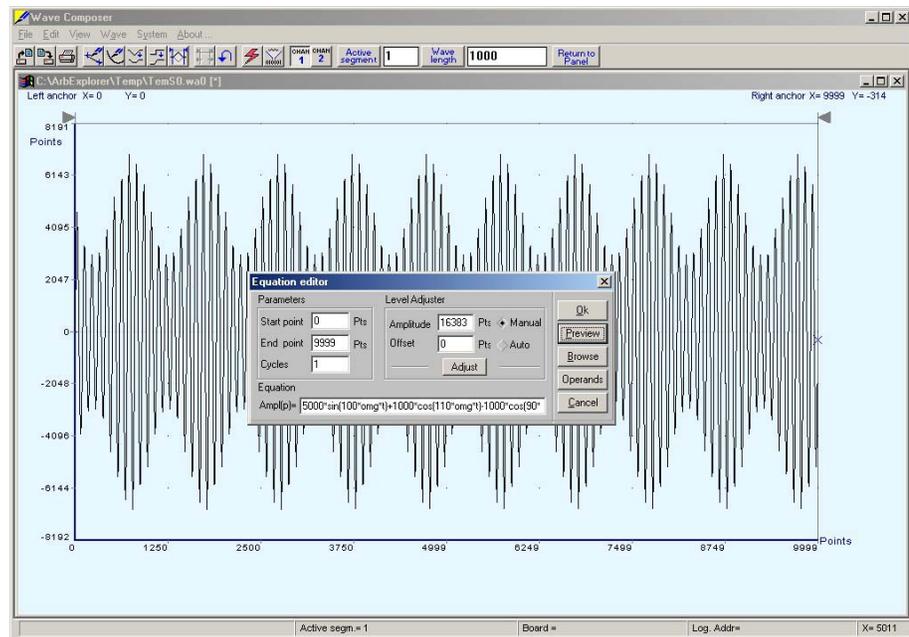
Figure 5-25. Using the Equation Editor to Build Amplitude Modulated Signal With Upper and Lower Sidebands

Combining Waveforms

The last but not least powerful feature allows you to combine waveforms which you previously stored on your hard disc. You can write mathematical expressions that contain waveforms, simple operands and trigonometric functions similar to the example given below. If you want to use waveforms in your equations, you must first generate these waves and store them on your hard disk. You identify waveforms by adding the *.wav extension as shown in the example below.

Ampl(p)= Sine.wav*sin(omg*p*10)*Noise.wav/1000

The above equation will generate amplitude-modulated waveform with added noise. The following steps demonstrate how to create, store and combine waveforms using this equation.

Step 1 – Create and store sine.wav. Invoke the Equation Editor and type the following equation:

Ampl(p)= 5000*sin(omg*p)

Press OK and then select the Save Waveform As… from the File command. Save this file using the name Sine.wav

Step 2 – Create and store Noise.wav. From the Wave command select Noise. Click OK and watch your waveform screen draw noisy signal. From the File menu select Save Waveform As… and save this waveform using the name Noise.wav.

Step 3 – Write and compute the original equation:

Ampl(p)= Sine.wav*sin(omg*p*10)*Noise.wav/1000

If you did not make any mistakes, your waveform screen should look as shown in Figure 5-26.
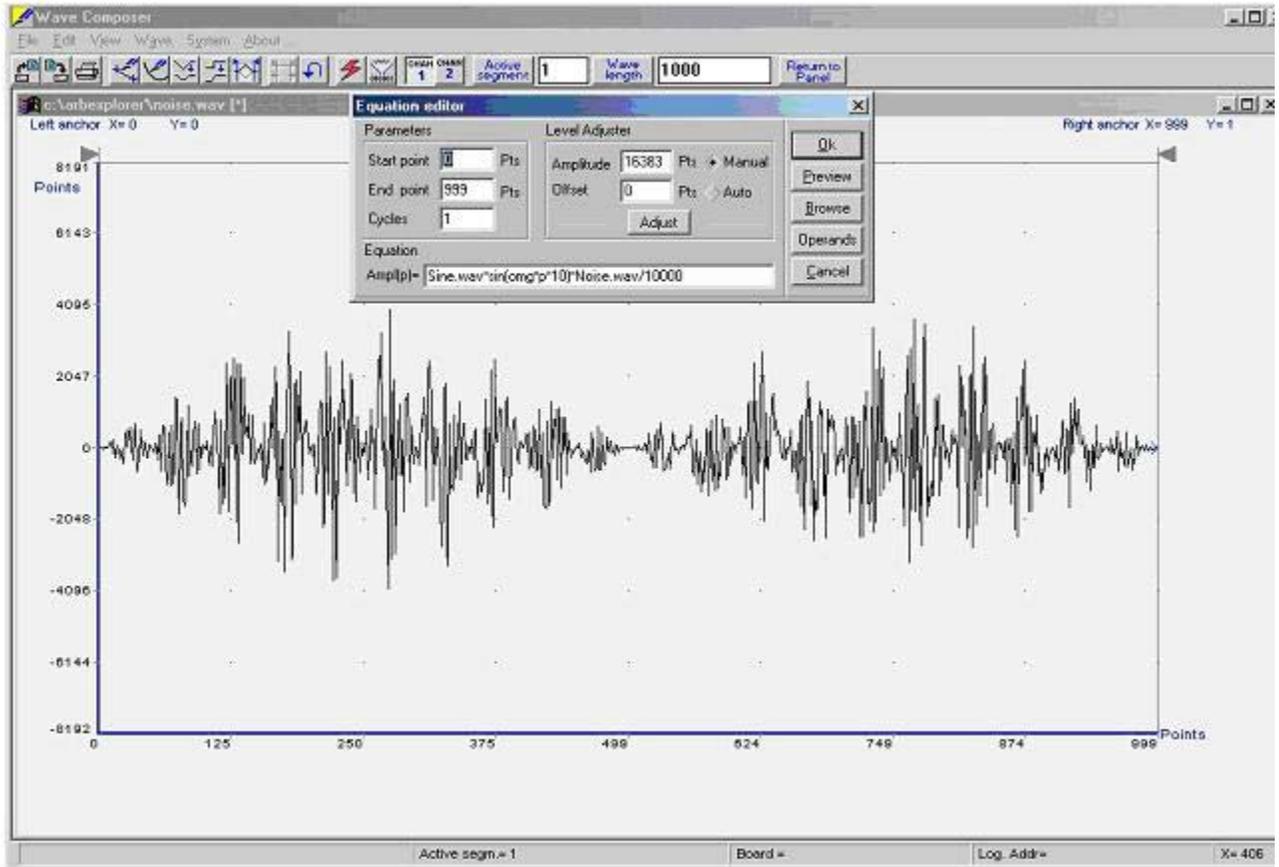
Figure 5-26. Combining Waveforms Into Equations

This page was intentionally left blank

Appendix A

SPECIFICATIONS

Configuration
Output Channels   2, semi-independent

Inter-Channel Control

Leading Edge Offset

Description	Channel 2 edge trails channel 1 edge by a programmable number of points.
Offset Units	Waveform points
Range	0 to 999999 points
Resolution and Accuracy	1 point, or 1 sample clock period of channel 2
Initial Skew	<±2 ns, with sclk divider = 1; <±3 ns, with sclk divider > 1

Channel 2 Sample Clock Divider

Description	The sample clock source is common to both channels 1 and 2. The sample clock for the slave channel can be divided down coherently from 1 to 64k.
Range	1 to 65,535
Resolution	1

Inter-Channel Dependency

Separate controls	Output on/off, amplitude, AM, offset, standard waveforms, user waveforms, user waveform size, sequence table, channel 2 clock divider, trigger start phase, breakpoints
Common Controls	Sample clock, frequency, reference source, trigger modes, sequence advance mode, SYNC output, FM, FSK, sweep, arm start/stop

Multiple Instrument synchronization

Description	Multiple instruments can be connected together and synchronized to provide multi-channel synchronization. Phase (leading edge) offset between master and slave units is programmable.

Leading Edge Offset

Description	Leading edge of master output trails the leading edge of the slave output by a programmable number of points. Each slave can be programmed to have individual offset
Phase Units	Waveform points
Range	0 to 999999 points
Resolution and Accuracy	4 point
Initial Skew	<±15 ns, depending on cable length and quality, typically with 1 meter coax cables

SAMPLE CLOCK

Sources:
Internal
Range	100 mS/s to 100 MS/s
Resolution	7 digits limited by 1 μS/s (10 digits using GPIB programming)
Accuracy	Same as reference
Stability	Same as reference
Reference	

Standard	$\geq$0.0001% (1 ppm TCXO) initial tolerance over a 19°C to 29°C temperature range; 1ppm/°C below 19°C and above 29°C; 1ppm/year aging rate
External	10 MHz TTL, 50% $\pm$2% duty cycle
External	From rear panel BNC, DC to 100 MHz
Level	ECL 100K compatible

SAMPLE CLOCK MODULATION

FM – Built-in Standard Waveforms
Description	Sample clock can be frequency modulated by internal waveforms that are resident in internal memory (fixed waveforms).
Modulation Source	Internal sine square, triangle and ramp
Modulation Frequency Range	1 mHz to 100 KHz
Resolution	7 digits
Accuracy	0.1%
Modulating Frequency Distortion	<0.1%
Deviation Range	100mS/s to 100 MS/s
Advance	Automatic, triggered, gated or software command
Marker	
Output and Level	Same as SYNC output.
Position	Programmable for selected frequency

FM – Downloaded Arbitrary Waveforms
Description	Sample clock can be frequency modulated arbitrary waveforms that are downloaded by the user (user waveforms.
Modulation Source	User waveform, any shape, 10 to 20000 waveform points
Modulation Sample Clock Range	1 mS/s to 2 MS/s
Resolution	7 digits
Accuracy	0.1%
Modulating Frequency Distortion	<0.1%
Sample Clock Deviation Range	100mS/s to 100 MS/s
Advance	Automatic, triggered, gated or software command
Marker	
Output and Level	Same as SYNC output.
Position	Programmable for selected frequency
Waveform Download	Through RS232, or DMA channel through GPIB

FSK

Description	Current segment is sampled continuously. External low TTL level programs carrier sample clock, external high TTL level programs shifted sample clock frequency. FSK operates in user shape (arbitrary) waveforms only.
Carrier Sample Clock Range	100 mS/s to 100 MS/s
FSK Source	
External	Rear panel Trigger input BNC. Low level = carrier sample clock; High level = hop frequency
Frequency Range	From 10 MHz to DC
FSK Delay	Minimum 1 waveform cycle + 50 ns

Ramped FSK

Description	Same as FSK except carrier sample clock ramps to shifted frequency at a rate defined by the ramp time parameter. External low TTL level programs carrier sample clock, external high TTL level programs shifted frequency.
Ramp Time Range	10 μs to 1 s, 3 digits, $\pm$0.1%

Sweep (*)

Description	Sample Clock sweeps continuously from start to stop, at a rate defined by the sweep time. More complex sweep modes and types can be generated using the FM mode in conjunction with the FM composer program.
Type	Linear or logarithmic
Direction	Up or down, depending on the start and stop setting
Range	100 mS/s to 100 MS/s
Time	1 ms to 1000 s, 7 digits, $\pm$0.1%
Advance	Automatic, triggered, gated or software command
Marker	
Output and Level	Same as SYNC output.
Position	Programmable for selected frequency

OPERATING MODES

Normal	Continuous waveform is generated
Triggered	Each input cycle generates a single output cycle.
Gated	External signal enables generator. First output cycle synchronous with the active slope of the triggering signal. Last cycle of output waveform always completed
External Burst	Preset number from 1 to 1M cycles stimulated by an external, or manual trigger. This mode is not available in SEQ mode.
Internal Burst	An internal timer repeatedly generates a burst from 1 to 1M counted cycles. This mode is not available in SEQ mode.

Trigger Sources

External	
Input	Front panel BNC
Level	TTL
Slope Sensitivity	Positive or negative, programmable
Frequency	2 MHz to DC
Internal	
Range	100 mHz to 2 MHz
Resolution	7 digits
Accuracy	0.1%
Software	IEEE 488.2 command

Trigger Start Phase

	Waveform starts from point n and completes at point n-1.
Programmable Units	Waveform points
Range	0 to 999999 waveform points
Resolution	4 points

Breakpoint

	Waveform starts after trigger and stops at breakpoint position. Each channel can be programmed with its unique breakpoints
Range	0 to 999999 waveform points
Event to cause stop at breakpoint	SCPI command or valid stop signal at the TRIG input.
Event to cause start from breakpoint	SCPI command or valid start signal at the TRIG input.
Breakpoint Error	±4 points

Start/Stop Control

Description	Valid signal at the Trigger input stops the waveform at a programmed breakpoint. The Stop signal operates simultaneously on both channels. Channel stop occurs separately and independently at a breakpoint uniquely programmed for each channel. If breakpoint has not been programmed, the waveform will stop after the last point.
Start/Stop control	Rear Panel Trigger Input BNC, RS232 or GPIB commands

System Delay

Trigger to waveform output	1 Sample Clock+150 ns

STANDARD WAVEFORMS

Frequency Range	Waveform dependent
Source Internal synthesizer	

Sine

Frequency Range	100 µHz to 50 MHz
Band Flatness	5% to 10 MHz; 20%, to 50 MHz
Programmable Parameters	Start phase, 0 to 360°

Triangle

Frequency Range:	100 µHz to 12.5 MHz, usable above 12.5 MHz
Adjustable Parameters	Start phase, 0 to 360°

Square

Frequency Range	100 µHz to 50 MHz
Adjustable Parameters	Duty cycle, 1% to 99%

Pulse/Ramp

Frequency Range	10 mHz to 12.5 MHz, usable above 12.5 MHz
Adjustable Parameters	
Delay	0% to 99.9% of period
Rise Time	0% to 99.9% of period
High Time	0% to 99.9% of period
Fall Time	0% to 99.9% of period

Sinc (Sine(x)/x)

Frequency Range	10 mHz to 12.5 MHz, usable above 12.5 MHz
Adjustable Parameters	
Cycles	4 to 100 cycles

Gaussian Pulse

Frequency Range	10 mHz to 12.5 MHz, usable above 12.5 MHz
Adjustable Parameters	

Time Constant	10 to 200

Exponential Decaying/Rising Pulse

Frequency Range	10 mHz to 12.5 MHz, usable above 12.5 MHz
Adjustable Parameters	
Time Constant	-100 to 100

DC

Range	-100% to 100% of amplitude

ARBITRARY WAVEFORMS

Waveform Memory	1 Meg points standard, 4 Meg points optional
Memory Segmentation	
Number of Segments	1 to 4096
Min Segment Size	16 points
Memory Interleave	4 (All trace lengths must be multiples of 4)
Vertical Resolution	14 bits (16,384 points)
Waveform Download	Through RS232, or DMA channel through GPIB

Sine Wave Performance

Description	Sine wave performance is measured using the maximum sample clock rate, at 5Vp-p, the maximum amplitude resolution of the DAC (14-bit) and without filters. Spectrum analyzer settings: Start frequency = 0MHz; stop frequency = 50MHz (Nyquist frequency)
Sinewave Total Harmonic Distortion	0.1% to 100 KHz
Harmonics and non-related spurious	
below 10MHz	< -55dBc for carrier frequencies $\leq$1MHz
	< -40dBc for carrier frequencies $\leq$5MHz
	< -35dBc for carrier frequencies $\leq$10MHz
	< -22dBc for carrier frequencies $\leq$50MHz

SEQUENCED ARBITRARY WAVEFORMS

Operation	Permits division of the memory bank into smaller segments. Segments may be linked, and repeated in user-selectable fashion to generate extremely long waveforms.
Advance Modes	
Automatic Sequence Advance	No triggers required to step from one segment to the next. Sequence is repeated continuously through a pre-programmed sequence table
Stepped Sequence Advance	Current segment is sampled continuously, external trigger advances to next programmed segment. Control input is TRIG IN connector.
Single Sequence Advance	Current segment is sampled to the end of the segment including repeats and idles there. Next trigger advances to next segment. Control input is TRIG IN connector.
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)
Advance Source	External, rear panel BNC; Internal; GPIB
Sequencer steps	From 1 to 4096
Segment loops	From 1 to 1 Meg
Segment Duration	Minimum 1 μs for more than one loop.

OUTPUTS

Channels 1 and 2 Output

Connector	Front panel BNC
Stand-by	Output Off or Normal
Impedance	50Ω, ±1%
Protection	Protected against temporary short to case ground
Amplitude	
Range	10 mV to 10 Vp-p, into 50Ω; Double into open circuit
Resolution	3.5 digits
Accuracy (1 KHz)	±(1% + 25 mV), 1.000 V to 10 Vp-p
	±(1% + 5 mV), 100 mV to 999.9 mVp-p
	±(1% + 2 mV), 10 mV to 99.99 mVp-p

Offset

Offset	Offset is attenuated with amplitude
Range	0 to ±4.5 V, amplitude dependent
Resolution	2.2 mV
Accuracy	±1%
Filters	50 MHz Elliptic
	25 MHz Elliptic

Square Wave, Pulse

Rise/Fall time	<10 ns, 10% to 90% of amplitude
Aberration	<5%

SYNC/Marker Output

Description	Provides dual functionality. All functions and modes, this output generates sync pulse, which is synchronous with the output waveform. In FM and sweep modes only, this output generates a marker at designated sample clock frequencies.
Connector	Front panel BNC
Impedance	50Ω, ±1%
Level	>2 V into 50Ω, 4V nominal into 10KΩ
Protection	Protected against temporary short to case ground
Validators	BIT, LCOM
Position	Point 0 to n, Programmable with 4-point resolution
Width Control	Programmable
Range	4 to 100000 waveform points
Resolution	4 points
Source	Channel 1

Sinewave Output

Description	An output that is directly derived from the sample clock generator and has an output frequency equal to the programmed sample clock frequency. This output generates sine waveforms that corresponds to sample clock setting, including modulated waveform, such as FM, sweep, FSK and Ramped FSK
Connector	Rear panel BNC
Impedance	50Ω, ±1%
Level	1 V into 50Ω
Protection	Protected against temporary short to case ground
Source	Sample clock frequency

Frequency Range and Resolution	Same as Sample clock
Harmonics and Spurious	
Total Harmonic Distortion	0.05% to 100 KHz
Harmonics and non-related	
spurious to 100 MHz	< 30dBc to 100 MHz

Multi-Instrument Synchronization Connectors

Description	These connectors are used for synchronizing one master instrument to multiple slave units, thereby creating a multi channel, fully synchronized and jitter free system. There are three connectors in this group, which must be connected between the instruments to achieve synchronization: SCLK output, SCLK input and 9-pin DSUB connector labeled Master/Slave.
Interconnecting Cables	Optional, consult factory at the time of purchase

SCLK Output
Connector	Rear panel BNC
Level	ECL into 50Ω, terminated to –2V

SCLK Input
Connector	Rear panel BNC
Input Level	ECL
Impedance	50Ω, terminated to –2V
Minimum Pulse Width	4 ns

Master/Slave
Connector	Rear panel 9-pin DSUB

INPUTS

TRIG Input
Connector	Rear panel BNC
Impedance	10 KΩ, ±5%
Threshold Level	TTL
Min Pulse Width	20 ns
Slope	Positive or negative going edge.

10 MHz Reference Input
Connector	Rear panel BNC
Impedance	10 KΩ, ±5%
Threshold Level	TTL
Duty Cycle	50%, ±5%

AM Input
Modulation Input	Rear panel BNC
Impedance	1 M, ±5%
Max Input Voltage	±12 V
Sensitivity	0 V to +5 V (5 Vp-p) produce 100% modulation
Source	External
Modulation Range	0 to 100%
Bandwidth	DC to 500 KHz

GENERAL

GPIB Information
GPIB Revision	IEEE-488.2
SCPI Revision	1993.0

Logical Address Settings	1 - 31, configured via internal DIP switch
DMA	Downloads arbitrary waveform data, arbitrary FM waveform data and sequence table data. DMA support is required by the controller

RS232 Information

RS232 Revision	RS232C
SCPI Revision	1993.0
Protocol	non-addressable

Front Panel Indicator LED's

Output On	Green – Output on / off (Separate for each channel)
SYNC On	Green – SYNC on / off

Power Requirements

Mains Input Range	85 to 265Vac, 48-440 Hz
Maximum Total Module Power	50W

Self Test Fault Coverage

Hardware Method	Internal relay wraps main output back to fault detection circuitry.
Fault Coverage (%)	90%

EMC Certification

CE marked

Mechanical

Dimensions	212 x 88 x 415 mm (W x H x D)
Weight	Approximately 6 Kg

Environmental

Operating temperature	0 °C - 50 °C	
Humidity (non-condensing)	11 °C - 30 °C	85%
	31 °C - 40 °C	75%
	41 °C - 50 °C	45%

Reliability

MTBF	per MIL-HDBK-217E, 25 °C, Ground Benign

Safety   EN61010

Appendix B

Controlling the Instrument via RS232

Introduction

It is possible to send SCPI command via HyperTerminal (or another serial communication utility) using the following setting:

Baud Rate: 115.2 K

Data Length: 8 bits

Parity: None

Stop Bit: 1

Flow Control: None (no hardware; no software)

Guidelines

When writing a program that works with RS232 you must add in each command buffer the query command *OPC? and wait until you receive response ('1NL') before sending next command. If, however, the command includes a query command, it is unnecessary to add the *OPC? query. Just wait for the response before sending the next command.

NOTES

The query response is NL terminated (0x0A) and no character is sent after NL (no 0xd and no NULL character)

When attempting to download waves or tables using a binary format (trace:data #.....; trace:seq#... or trace:seg....#) the following protocol must be used:

1. Send command buffer: "*opc?;:trace:data#" with nothing after #(no NL or 0)

2. Wait for opc response ("1NL")

3. Send number of byte in size (one ASCII char between 2 to 9)

4.   Wait for character '1' and then send ASCII for number of bytes in wave (number of points *2)

- (if receive '0' with syst:err? query it's possible to know what error find)

5.   Wait for character '1' and then send binary data as explained in the manual. After sending the firmware, send '1NL' and return to panel mode. (If receive '0' with syst:err? query it's possible to know what error find)

NOTES

Use the same protocol sequence for trace:seq# and trace:seg#

REPAIR AND CALIBRATION REQUEST FORM

To allow us to better understand your repair requests, we suggest you use the following outline when calling and include a copy with your instrument to be sent to the Racal Repair Facility.

Model_____ Serial No._____ Date_____

Company Name_____ Purchase Order #_____

Billing Address_____

City

State/Province   Zip/Postal Code   Country

Shipping Address_____

City

State/Province   Zip/Postal Code   Country

Technical Contact_____ Phone Number (   )_____
Purchasing Contact _____ Phone Number (   )_____

1. Describe, in detail, the problem and symptoms you are having. Please include all set up details, such as input/output levels, frequencies, waveform details, etc.

_____
_____
_____
_____

2. If problem is occurring when unit is in remote, please list the program strings used and the controller type.

_____
_____
_____

3. Please give any additional information you feel would be beneficial in facilitating a faster repair time (i.e., modifications, etc.)

_____
_____
_____

4. Is calibration data required?   Yes   No   (please circle one)

Call before shipping   Ship instruments to nearest support office
Note: We do not accept   listed on back.
"collect" shipments.