5040

Timer/Counter
& Oscilloscope Calibrator

User Manual

Document History

Version	Date	Description
1.0	14/11/01	Initial version

Contents

5040 CONFIGURATION

Communication Settings

The 5040 can communicate via GPIB or RS232 (serial) connection – GPIB is the preferred option, since more than one device can be connected on the same bus. It also requires less configuration on the PC (i.e. it does not require baud rate or parity to be selected) so is generally less troublesome.

If RS232 is selected, the protocol is fixed at 9600 baud, 8 data bits, no parity, and 1 stop bit.

Comms settings are set using the DIP Switch on the back panel.

DIP Switch Settings (back panel)

DIP Switch	Use
1	GPIB/Serial – OFF=GPIB (default), ON=Serial
2-6	Binary-coded GPIB address (not used if DIP1 is ON). When the switches are OFF, they have the value 0. When the switches are ON, they have the following values: Switch 2 = 1 Switch 3 = 2 Switch 4 = 4 Switch 5 = 8 Switch 6 = 16 The GPIB address is given by the **sum** of the switch values. For example, to select GPIB address 20, put switches 4 (value 4) and 6 (value 16) in the ON position and put switches 2, 3, and 5 in the OFF position.
7-8	Not used – Keep in the OFF position

Only change the DIP switch settings while the unit is turned off – changes in DIP switch settings will not be recognized while the unit is on.

LED Indicators (front-panel)

The LED indicators on the front-panel are used to indicate the status of the unit:

LED	Use
Power	When on, the unit is receiving mains power.
GPIB	When on, the unit is configured to use GPIB comms
RS232	When on, the unit is configured to use RS232 comms
Data	Flashes when a command is received over GPIB/RS232
Amplitude	Lights when unit is in Amplitude Mode
Freq/Period	Lights when unit is in Frequency or Period Mode

Connections

DC Volts Connection

The 5040 makes use of the DC Volts output from a 5022 (or 982x) when in Amplitude Mode. The DCV Input terminals of the 5040 (on its back panel) must therefore be connected to the DCV Output terminals of the 5022.

Before making this connection, ensure that both the 5040 and 5022 are switched off.

Communication Connections

The 5040 must be connected to the PC by a GPIB cable (or a “straight-through” RS232 cable if RS232 is being used).

In addition, the 5022 must also be connected to the PC by a GPIB cable.

Ensure the 5022 is switched to Remote operation on its Remote/Local switch. If this is not done, the 5022 will not be controllable by the PC.

5040 MANUAL CONTROL APPLIACTION

The 5040 Manual Control Application runs under Windows 95/98/Me/NT4/2000 and XP. It provides a simple interface to the 5040 allowing all its features to be accessed without programming.

To start the application, locate the **5040 Manual Control icon** on your Desktop, or start it using the Windows *Start* button: **Programs->Time Electronics 5040->5040 Manual Control**.

Configuring the Manual Control Application

The Manual Control App must be told how the 5040 and the 5022 (or 982x) are communicating. After starting the 5040 Manual Control App, click the **Setup** button. Select how the 5040 will communicate with the PC – either GPIB or RS232. If GPIB you must also indicate which GPIB address the 5040 is configured to.

You must also tell it the GPIB address of the Voltage Source Calibrator (i.e. 5022 or 982x). Having done this, click Test/Reset to test the settings. Then click OK to save the settings and exit the Setup window. The settings will be saved so you should not need to repeat this operation unless you need to change the communication options of the 5040 or 5022.

Features of the Manual Control Application

The Manual Control App features a single screen from which all the features of the 5040 can be selected.

The top-left of the screen shows the present output settings.

Below this on the left-hand side are the main options:

Function:	Frequency/Period/Amplitude/Fast Rise
Range/Value:	Shows possible ranges and fixed values for the chosen function
Graticule Height:	When in amplitude mode, this selects how many y-axis graticules the amplitude test will occupy, e.g. 6

The numeric keypad allows the selection of an output value. Key in the value required then click the appropriate button below the numeric keypad, e.g. to set 250kHz, click **“2”, “5”, “0”. “kHz”**.

In addition, frequencies (200kHz and slower) and periods (4us and longer) can have their duty cycle varied. This is done by keying in the required duty cycle (0.001 to 99.999) then clicking the **“Duty Cycle”** button.

In Amplitude Mode, the amplitude output can be deviated in small steps. This is done using the Deviation buttons in the top-right corner of the screen. Deviation mode is turned on and off by clicking the **“Deviation”** button. When Deviation is on, the buttons allowing coarse and fine deviation arte enabled.

Fast Rise mode outputs a set 20ns waveform. By entering the rise time observed on the oscilloscope under test, the application can calculate its bandwidth.

PROGRAMMING GUIDE TO THE 5040

Introduction

The 5040 features a small set of commands that allow control of its output. These commands can be sent to the 5040 by GPIB or RS232 serial.

In addition, commands are also sent via GPIB to the Voltage Source (5022 or 982x) in order to control the DC Volts going into the 5040. If using EasyCal, these 5022 (or 982x) are performed automatically.

This guide describes how to control the 5040 via GPIB/RS232. It also includes example EasyCal Tests showing the commands in use. A reference guide listing all the commands is given at the end of this guide.

Command Format

All commands are case-insensitive, i.e. you can use upper/lowercase letters in any combination you like.

All commands must be terminated, either by a signal on GPIB's EOI line (this is automatically sent by most GPIB interface boards), or a linefeed character (ASCII code 10), or a carriage return character (ASCII code 13).

In addition, a 100 millisecond pause after each command is required before sending another command – if this is not done then the following command may be missed.

Sending GPIB Commands from EasyCal

EasyCal features extra options when sending GPIB commands that are useful when controlling the 5040:

- Ability to send more than one command in a single line
- Ability to delay sending of a command

Both these features are illustrated by this example of a "GPIB Before" command in an EasyCal test:

```
f100000{13}{wait100ms}d25
```

EasyCal interprets this command and performs the following tasks:

1. Sends command `f100000` to set the 5040 output to 100kHz
2. Tells EasyCal to then send character ASCII code 13 (carriage return) as that command's terminator, allowing another command to be put afterwards
3. Tells EasyCal to waits 100 milliseconds before sending the second command – this allows the 5040 time to process the previous command. Without the delay, the following command might be missed by the 5040.
4. Sends the command `d25` to set the duty cycle to 25%.

Note: No terminating char is required after the *last* command, `d25,` when using EasyCal since EasyCal sends the required terminating char/EOI signal by default.

Configuring EasyCal to Control the 5040

THIS PROCEDURE WILL HAVE ALREADY BEEN DONE IF EASYCAL WAS SUPPLIED AT THE SAME TIME AS THE 5040

You need to tell EasyCal that you have a 5040 on your system. To do this it must be added to EasyCal's list of Calibration Instruments:

1. Start **EasyEdit**
2. Select **System Setup** then **Calibration Instruments**
3. Look at the list of calibration instruments: Is there one with "5040" in its description? If not, then you need to add it:
    a. Click the **Add…** button
    b. Select **Type** as *GPIB/RS232 Instrument*
    c. Enter the **Description** as *5040 Timer/Counter & Scope Cal* (or use your own wording)
    d. Select **Comms** options, recommend are: *GPIB* with address *10*. If you are using RS232, then select *RS232* with *1,9600,8,n,1*. The Comms options selected must agree with the DIP switch settings on the 5040.
    e. Enter the **Serial Number** and **Certificate Number** of the 5040 into the boxes.
    f. Leave the **Supports Readback** box unchecked
    g. Click OK to save the information and return to the list of calibration instruments
    h. Exit the Calibration Instruments window back to the menu

Controlling the 5040 from EasyCal

Commands can be sent to the 5040 from any test in a procedure. When adding a new test to a procedure you must first decide what type of test is needed.

The type of test to add depends on what you want the 5040 to do.

The jobs that the 5040 can do fall into these categories:

(a) Initialize itself (required before any other 5040 tasks can be performed)

(b) Perform a Frequency test

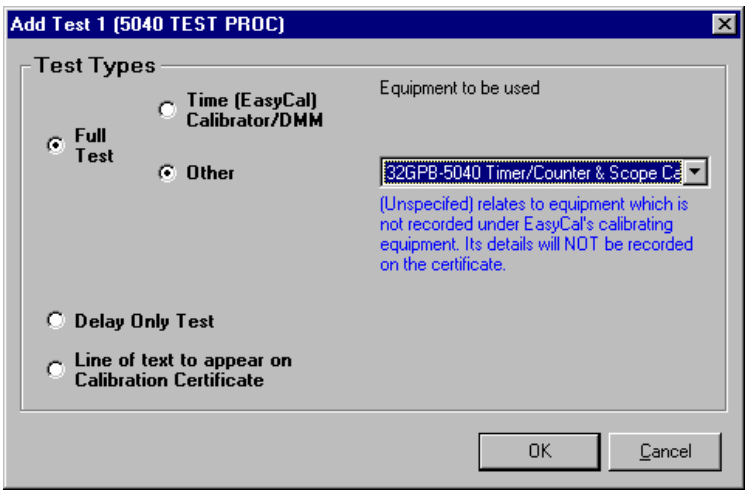
(c) Perform a Period test

(d) Perform a Duty Cycle test

(e) Perform an Amplitude test

Category (a)-(d)

These types of tasks all require the same type of test to be added. Select the Test Type as shown in this screenshot:



The **Other** option is selected, to allow selection of a Calibration Instrument.

The **5040** is then selected from the pull-down box as the *Equipment to be Used*.

Category (e) - Amplitude

The type of test required for Amplitude tests is different to all the others, since it also requires control of the 5022. Amplitude tests are covered further on in this guide.

Initializing the 5040

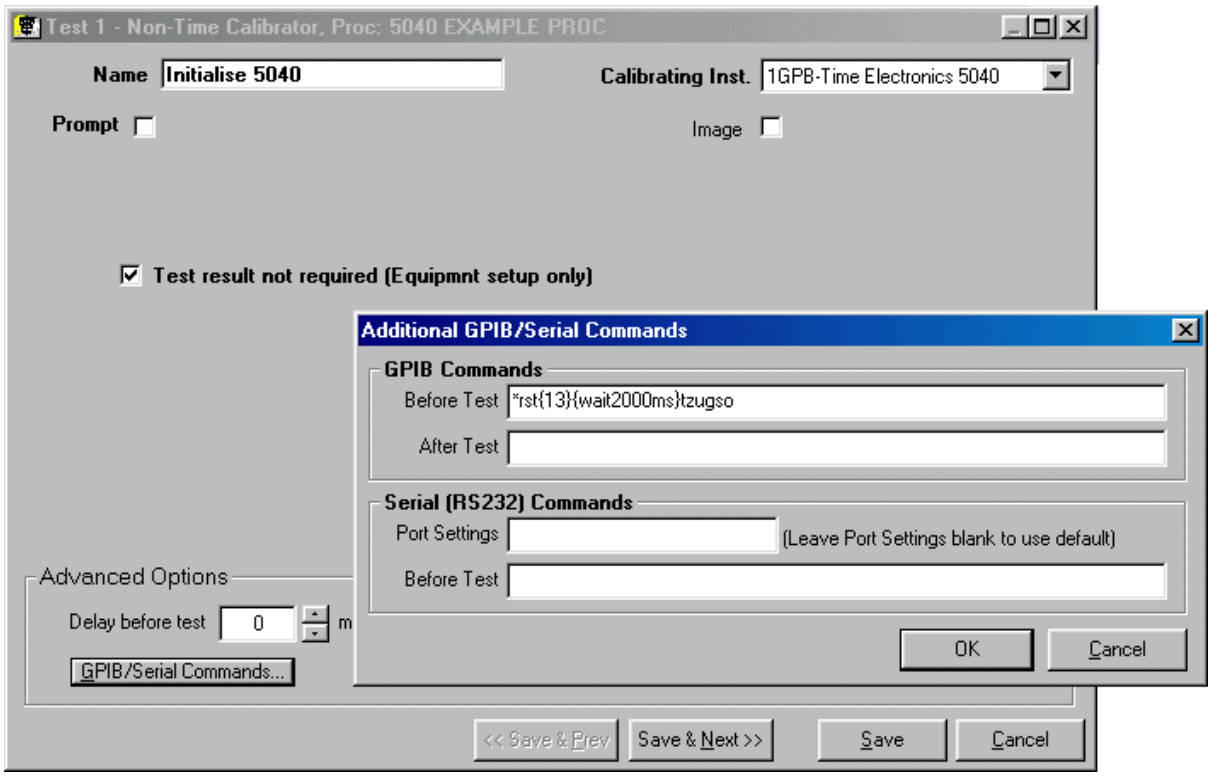
Each time the 5040 is turned on or reset, it must be initialized before any commands will be accepted. The command to initialize the 5040 is:

```
tzugso
```

This command does not need to be sent again unless the 5040 is turned off or reset via a GPIB/RS232 command.

Example EasyCal Test to Initialize 5040

Any procedure that controls the 5040 should include a test at the beginning to initialize the 5040. Select the test type as discussed above. Then fill in the test details as shown here:



Note that the **Test result not required** box is checked. This means that no result is required from the test and nothing relating to this test will be printed on the Results Certificate – this type of test is sometimes known as a *"setup-only"* test, since it is simply used to configure a piece of equipment prior to a test itself.

The **Before Test GPIB Command** contains the commands sent to the 5040:

`*rst{13}`	*Sends the GPIB reset command to reset the 5040 to its power-up defaults. The {13} sends a termination char to the 5040 (required since there is another command following)*
`{wait2000ms}`	*Causes EasyCal to wait 2 seconds (2000ms) before sending the next command – required to allow time for the 5040 to reset.*
`tzugso`	*Sent to the 5040 causing it to initialize and be ready to accept further commands.*

Using the 5040 in Frequency/Period Mode

Frequency Output

The 5040 can be set to output any of these frequencies:

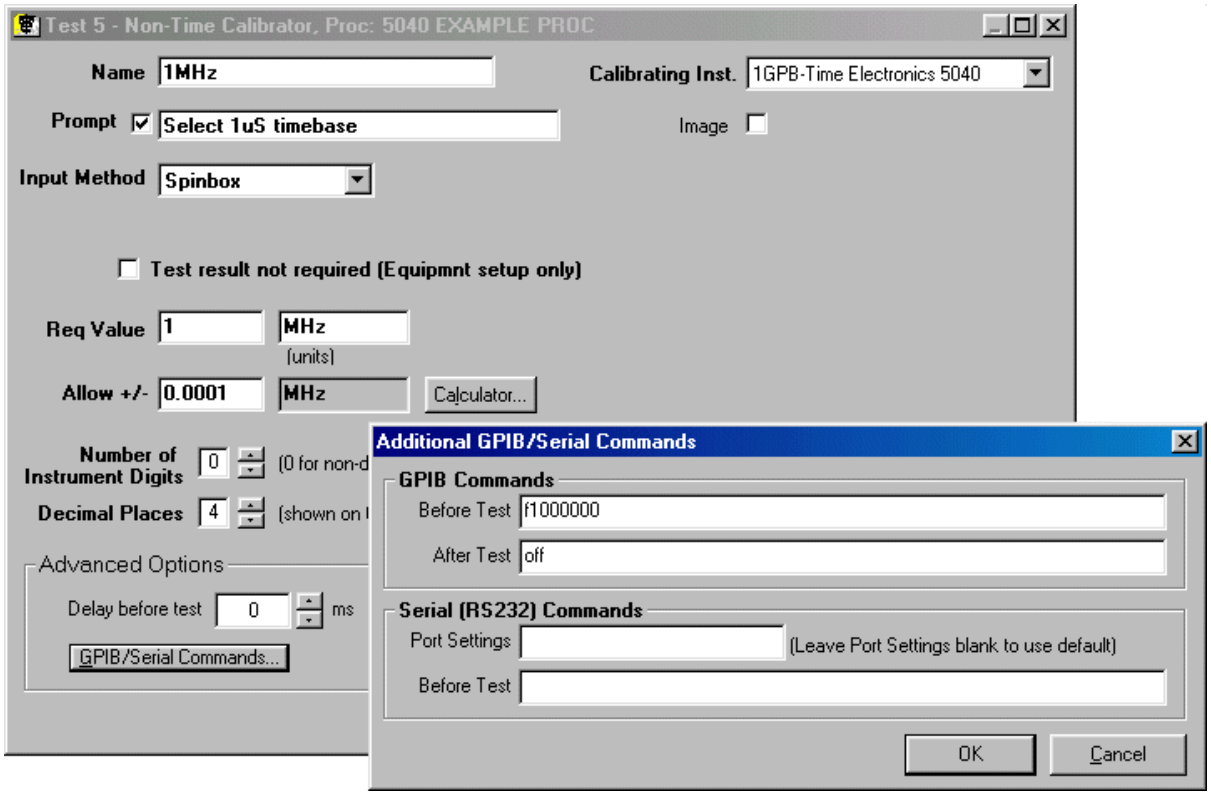
- 0.2Hz to 200kHz *(settable in steps equivalent to 1us intervals)*
- 500kHz
- 1Mhz
- 2MHz
- 2.5MHz
- 4MHz
- 5MHz
- 10MHz
- 20MHz
- 25MHz
- 50MHz
- 100MHz

To set any of these frequencies, the command **`f`** is used followed immediately by the frequency specified in hertz, e.g.:

`f0.1`	*0.1Hz*
`f58.6`	*58.6Hz*
`f1234.5`	*1.2345kHz*
`f200000`	*200kHz*
`f100000000`	*100MHz*

Example EasyCal Frequency Output Test

This screenshot shows an example of how to create a test to output a frequency of 1MHz:



Fill in the **Required Value** with the frequency and its units. Also fill in the **Allow +/-** box (it will use the same units).

In order to tell the 5040 what frequency to output, a **Before Test GPIB Command** is used, containing a command sent to the 5040:

`f1000000`	*Tells the 5040 to enter Frequency Mode and set its frequency to 1MHz (1,000,000 Hz)*

In addition, there is also an **After Test GPIB Command**: `off` – this turns off all output from the 5040 – it is good practice to do this in all types of test.

Period Output

The 5040 can be set to output any of these periods:

- 10ns
- 20ns
- 40ns
- 50ns
- 100ns
- 200ns
- 250ns
- 400ns
- 500ns

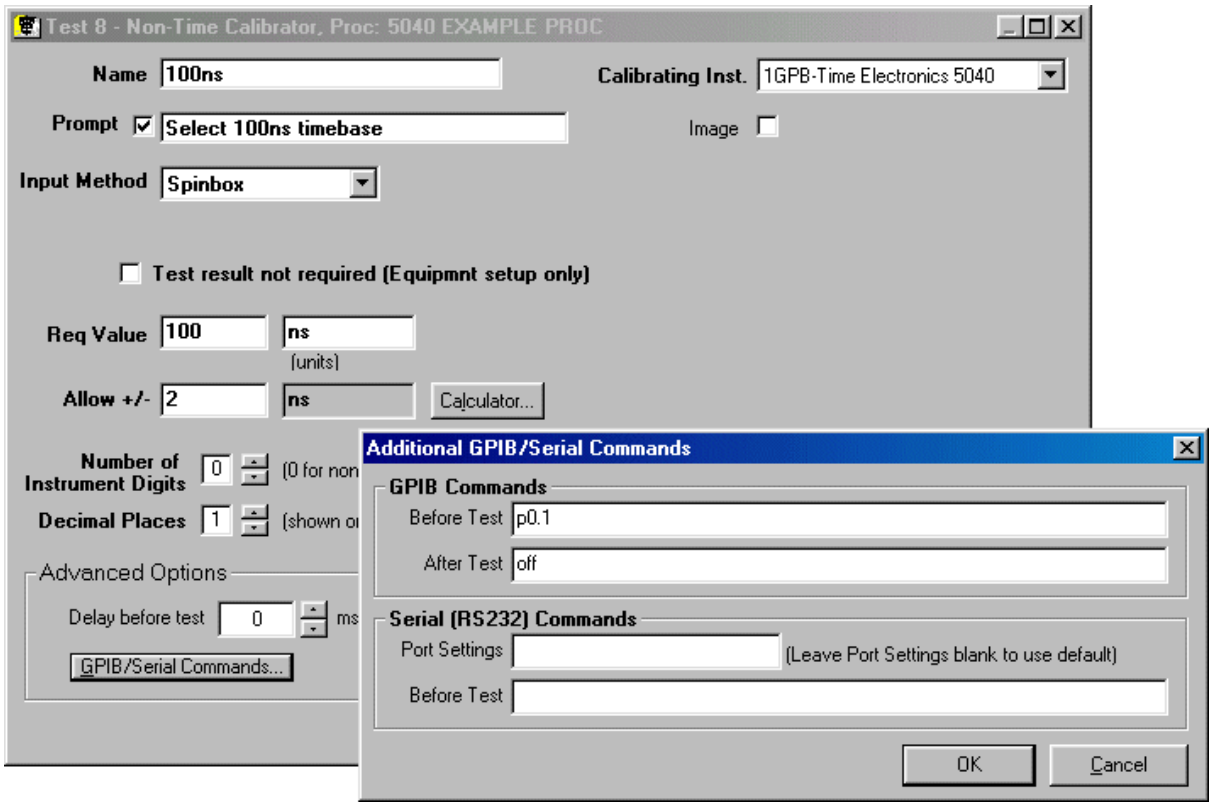
- 1us
- 2us
- 4us to 8s *(settable in 1us steps)*

To set any of these frequencies, the command **`p`** is used followed immediately by the period specified in microseconds, e.g.:

`p0.01`	*10ns*
`p0.2`	*200ns*
`p5`	*4us*
`p1000`	*1ms*
`p1000000`	*1s*

Example EasyCal Period Output Test

This screenshot shows an example of a test to output a frequency of 100ns:



Fill in the **Required Value** with the period and its units. Also fill in the **Allow +/-** box (it will use the same units).

In order to tell the 5040 what period to output, a **Before Test GPIB Command** is used, containing a command sent to the 5040:

`p0.1`	*Tells the 5040 to enter Period Mode and set its period to 100ns (0.1us)*

The **After Test GPIB Command** (`off`) turns off all output from the 5040.

Duty Cycle

Duty cycle output for frequencies above 200kHz (or periods less than 4us) is fixed at 50% (except at 2MHz [500ns] where it is ~48%, and 20MHz [50ns] where it is ~40%).

For frequencies of 200kHz or slower (or periods of 4us or longer), the duty cycle can be varied from 50% by sending a command.

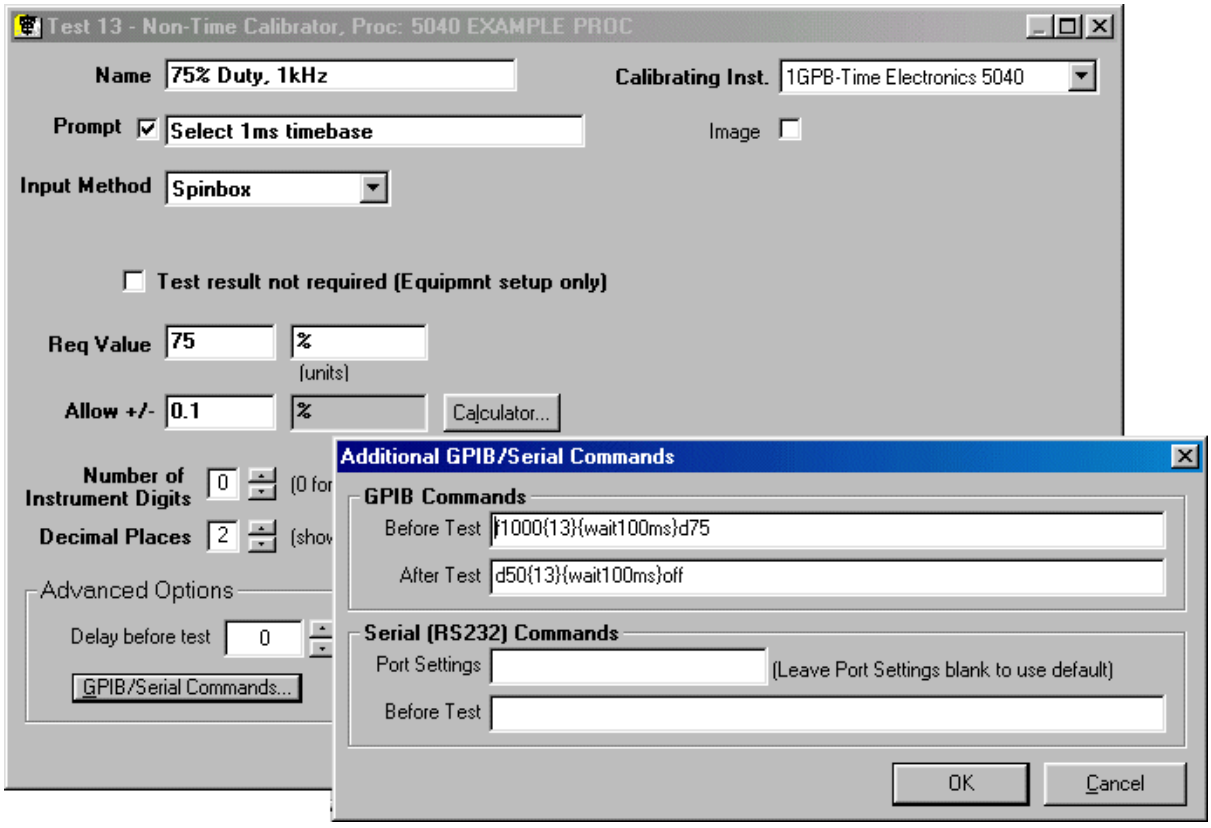
To set the duty cycle send `d` followed by the duty cycle, e.g.:

`d25`	*25%*
`d3.333`	*3.333%*
`d66.667`	*66.667%*
`d99.9`	*99.9*
`d50`	*50% (default)*

Note, since the duty cycle can only vary the on/off time in steps of 1us, at faster frequencies (and shorter periods), the step size of the duty cycle is much reduced – the closest value to the duty cycle requested will be selected.

Example EasyCal Duty Cycle Output Test

This screenshot shows an example of a test to output a duty cycle of 75% at 1kHz:



The **Req Value** is entered as the duty cycle being output, and so its units are **%**.

The 5040's output is configured by the **Before Test GPIB Command**:

f1000{13}	*Puts the 5040 into Frequency Mode and sets its frequency to 1kHz. The {13} is required to terminate this command so that another command can follow on the same line.*
{wait100ms}	*Causes EasyCal to wait 100ms before sending the next command – required to allow time for the 5040 to carry out the previous command.*
d75	*Sets the duty cycle of the 5040's output to 75%.*

In addition, there is also an **After Test GPIB Command**:

d50{13}	*Returns the duty cycle to the 50% default for any test that follows this one.*
{wait100ms}	*Causes EasyCal to wait 100ms before sending the next command – required to allow time for the 5040 to carry out the previous command.*
off	*Turns off all output from the 5040.*

Amplitude Output

Amplitude Output is achieved by a combination of the Voltage Source (5022 or 982x) and the 5040. The Voltage Source provides the precision DC voltage and 5040 then processes it, e.g. passing it through a buffer to divide it down, or by switching it to create a 1kHz square wave output.

In order to produce Amplitude Output then, commands must be sent to both the 5040 and the 5022.

Fortunately, by using EasyCal, the control commands for the 5022 (or 982x) are handled automatically. Only the commands to control the 5040 must be entered into EasyCal.

Use of Buffers

The 5040 has 4 voltage buffer selections:

- No buffer
- x1 buffer
- ÷10 buffer
- ÷100 buffer

The buffers are used to scale the input voltage coming from the voltage source (i.e. 5022), so that it can keep its output in the most accurate range as possible. E.g. to produce an output of 600mV, the voltage source is set to output 6V and the ÷10 buffer is selected, producing an amplitude of 600mV from the 5040.

Different buffer selections are made depending on the output voltage level required.

In order to calculate what buffer setting to use for any given required Amplitude, follow these steps:

1. Calculate the **Total Amplitude** required from the 5040:

   5040 Total Amplitude = ***Amplitude per Graticule*** x ***Graticule Height***

   e.g.: For 100mV per Graticule and 6 Graticules Height, the 5040 Total Amplitude is 600mV.

2. Using the 5040 Total Amplitude value, refer to this table to work out which 5040 Buffer Command must be sent to the 5040:

5040 Total Amplitude	5022 DCV Output	5040 Buffer Command
up to 80mV	0-8V	**a3** (÷100 buffer)
up to 800mV	0-8V	**a2** (÷10 buffer)
up to 8V	0-8V	**a1** (x1 buffer)
up to 400V	0-400V	**a0** (unbuffered)

   *e.g.: If the **5040 Total Amplitude** is 600mV, then we will need to send the **a2** command to the 5040 to select its ÷10 buffer.*

3. Calculate what the 5022's DCV output must be in order to produce the required amplitude, after taking account of any divide-by buffers used on the 5040.

   e.g.: If the 5040 will be using its ÷10 buffer, then the output from the 5022 must be 10 x the 5040 Total Amplitude. So if 5040 Total Amplitude is 600mV, the output of the 5022 must be 6V.

DC/AC Output Selection

The 5040 can output Amplitude either as an AC signal (square wave at approximately 1kHz frequency) or as DC. The selection is done using these commands:

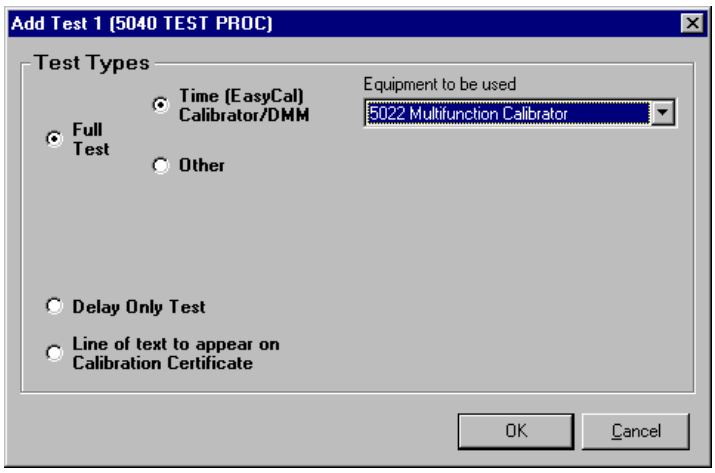
w0 *(DC output)*

w1 *(AC output)*

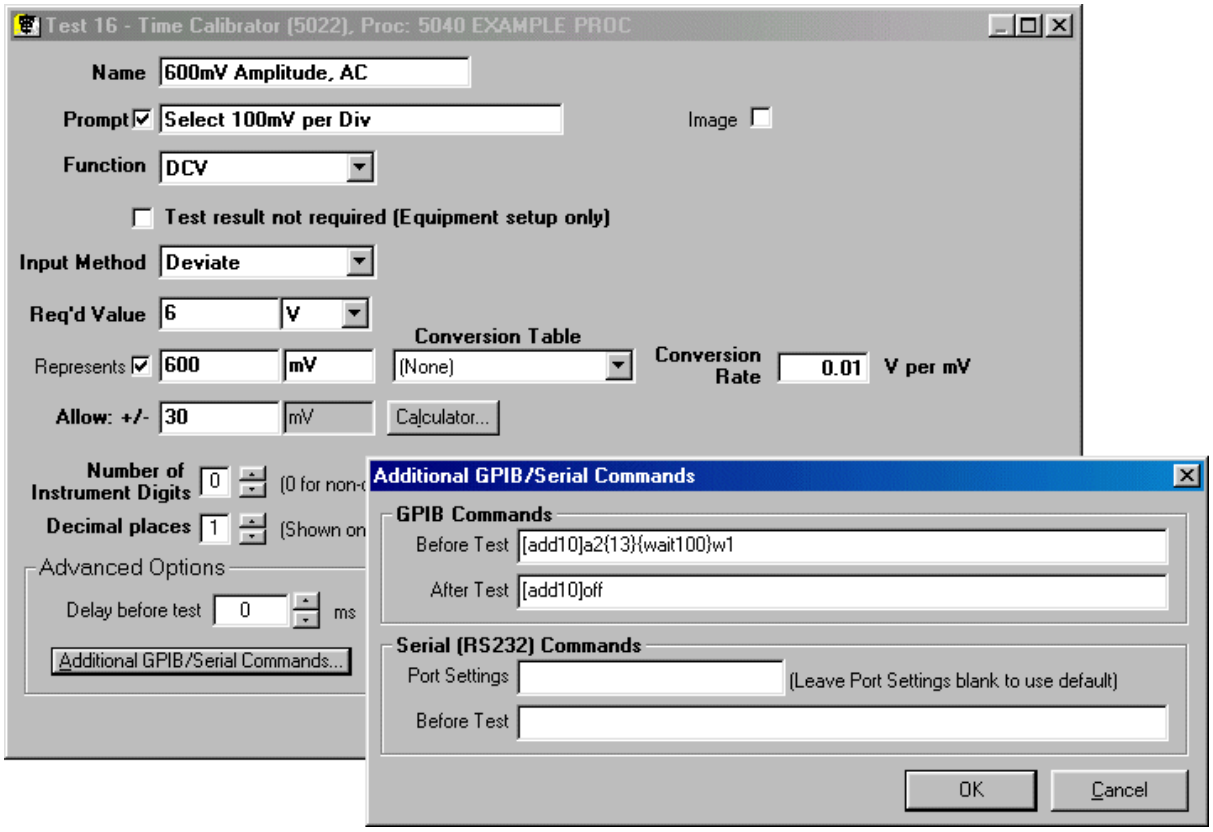
Example EasyCal Amplitude Output Test

When performing an Amplitude test, both the 5022 and the 5040 require control. For this reason, the test type selected must be a 5022 test. The 5040 is then controlled with some additional GPIB/RS232 commands.

When adding an Amplitude test then, select the **Type of Test** as shown here:


Add Test 1 (5040 TEST PROC)
Test Types
Full Test
Time (EasyCal) Calibrator/DMM
Other
Equipment to be used
5022 Multifunction Calibrator
Delay Only Test
Line of text to appear on Calibration Certificate
OK
Cancel

The test details are then entered. This screenshot shows an example of a test for performing a 600mV Amplitude test (1kHz AC), i.e. 100mV per graticule across 6 graticules:



Important things to note are:

Function — *Must be **DCV** since 5040 always requires DC Volts from the 5022, regardless of whether the test is AC or DC.*

Req'd Value — *This is always entered as the voltage that the 5022 will output. THIS IS NOT THE SAME AS THE AMPLITUDE FROM THE 5040 IF A DIVIDE-BY BUFFER IS BEING USED ON THE 5040.*

Represents — *In tests where the 5022's output voltage goes through the 5040's x1 buffer or is unbuffered, the Represents line should not be used so leave the **Represents** box unchecked.*

In this example, however, the 5022's output voltage is divided down by the 5040. If this is the case the Represents line must be used in order to display the correct values during the test run and on the Results Certificate.

The value entered on the Represents line is the voltage output by the 5040, in this case 600mV, since we will be using the 5040's ÷10 buffer.

Conversion Table — *Leave at [none]*

Conversion Rate *This value tells EasyCal that 1mV change in the out from the 5040 will occur for every 10mV (i.e. 0.01V) of change of the output from the 5022.*

Use these figures:

*When using the **÷10 buffer**, the rate will be **0.01V per mV***
*When using the **÷100 buffer**, the rate will be **0.1V per mV***

As with all other 5040 tests, a **Before Test GPIB Command** is used to control the 5040:

[add10]	*Not a command, rather an instruction to EasyCal to send the following GPIB commands to GPIB address 10 (i.e. the 5040). This is required in a 5022 test since otherwise the GPIB commands will be sent to the 5022.*
a2{13}	*Tells the 5040 to select ÷10 buffer.*
{wait100ms}	*Allows time for the 5040 to complete the previous command.*
w1	*Tells the 5040 to select the AC (1kHz) output waveform*

Again, an After Test GPIB Command is used:

[add10]off	*Sends "off" to GPIB address 10, i.e. the 5040, to tell it to stop output.*

More EasyCal Examples

An example procedure (**5040 EXAMPLE PROC**) is included with EasyCal.

The tests in that procedure cover all the different types of test that can be performed with a 5040. Take a look at a test in the example procedure that is close to what you want to do – remember to look at the **GPIB/Serial Commands** as well as the main test details.

COMMAND REFERENCE

Allow a gap of 100ms after each command before sending the next command.

Command	Parameter	Use	Example
tzugso	-	Initializes the 5040 ready to accept commands. This command MUST be sent before any other, otherwise the unit will ignore those commands. If the 5040 is reset or turned off, then this command must be sent again.	**tzugso**
off	-	Turns off all output from the 5040. This is the power-up default state.	**off**
Frequency/Period Mode commands			
f	Hertz	Sets the 5040 output to Frequency Mode and sets the output in Hertz. Valid frequencies are as shown in the Frequencies Table.	**f250000**
p	microseconds	Sets the 5040 output to Period Mode and sets the output in microseconds. Valid periods are as shown in the Periods Table.	**p100**
d	duty cycle %	Sets the duty cycle percentage of a frequency/period waveform. The 5040 must be in Frequency/Period Mode before calling this function. Duty Cycle can be between 0.1 and 99.9	**d81.1**
Amplitude Mode commands			
a0	-	Select Amplitude Mode, unbuffered. See Amplitude Mode table for info.	**a0**
a1	-	Select Amplitude Mode, applying a times 1 buffer to the input voltage. See Amplitude Mode table for info.	**a1**
a2	-	Select Amplitude Mode, applying a divide by ten buffer to the input voltage. See Amplitude Mode table for info.	**a2**
a3	-	Select Amplitude Mode, applying a divide by hundred buffer to the input voltage. See Amplitude Mode table for info.	**a3**
w0	-	Set output to DC. Must be in Amplitude Mode before sending this command.	**w0**
w1	-	Set output to 1kHz AC. Must be in Amplitude Mode before sending this command.	**w1**

Command	Parameter	Use	Example
Other Commands			
t	-	Transmits the current setting of the 5040. If GPIB comms is being used, then this command must be followed by a GPIB “receive” before any more commands will be accepted.	**t**
o1	-	Sets the terminating character of comms sent **by** the unit to a linefeed (ASCII code 10).	**o1**
o2	-	Sets the terminating character of comms sent **by** the unit to a carriage return (ASCII code 13).	**o2**
o12	-	Sets the terminating character of comms sent **by** the unit to a linefeed (ASCII code 10) followed by a carriage return (ASCII code 13).	**o12**
o21	-	Sets the terminating character of comms sent **by** the unit to a carriage return (ASCII code 13) followed by a linefeed (ASCII code 10).	**o21**
o	-	Sets the terminating character to nothing. Note, if using GPIB then comms will always terminate in the EOI signal.	**o**
g1	-	Enter G.E.T. mode (only available in GPIB)	**g1**
g2	-	Leave G.E.T. mode	**g2**
IEEE488.2-Compliance Commands (GPIB only)			
***trg**	-	G.E.T. trigger	***trg**
***rst**	-	Resets the unit to its power-up state. Allow 1 second afterwards before sending another command	***rst**