

June 4, 2010

PinPoint Monitor Software Manual
Version 4.4

GEC Instruments Precision Temperature Measurement,
Display, Graphing and Logging System

GEC Instruments
5530 N. W. 97th Street
Gainesville, FL 32653

Table of Contents

SOFTWARE INSTALLATION.....	1
HARDWARE FILES	2
RUNNING THE SOFTWARE.....	2
THE MENU BAR.....	3
THE MAIN TOOLBAR	3
THE STATUS BAR.....	4
THE REAL-TIME REPORT	5
The Aliases	5
THE REALTIME GRAPH	6
DATA LOGGING	7
RETRIEVING DATA FROM A PPM DATABASE.....	9
HARDWARE SETTINGS.....	10
CALIBRATION PROCEDURE.....	11
ALARMS	13
CONFIGURING DIFFERENTIAL THERMOCOUPLE INPUTS	14
CLOSING THE PROGRAM.....	14
THE PPM DYNAMIC DATA EXCHANGE SERVER - DDE	14
REALTIME GRAPH PLOTTING OPTIONS	16
TROUBLESHOOTING PROCEDURES.....	21
PLOTTING FROM A PPM DATABASE WITH PINPOINT VIEWER.....	23
USING CUSTOM THERMOCOUPLE COEFFICIENTS WITH TCCONFIG.....	25

SOFTWARE INSTALLATION

The GEC PinPoint Monitor software is designed to run on Windows NT, Windows 2000, or Windows XP. Most of the software features will function normally under W95, W98, or Windows ME, but these operating systems are no longer being supported.

The software will also run under Windows Vista, but with reduced performance. To avoid intermittent program lockup when running on Vista, go to the Task Manager, open the Processes Tab, highlight PPMonitor.exe, right click, and set the priority to Above Normal. This operation must be performed each time you restart PPM on Vista.

System requirements are 128k of memory and a 500Hz or higher processor.

Four programs are provided with your GEC instrument:

PinPoint Monitor (PPM) is the main program used for display, plotting and logging of temperatures and/or other inputs (such as relative humidity) that are available in your hardware.

PinPoint Settings (PPS) is used to set the Com Port, the default temperature units, and the configuration file for PPM. Com Port entry is a text field to allow for custom or higher numbered Com Ports.

PinPoint Viewer (PPV) is used to view and export data from logged files.

PinPoint Parser (PPP) is used to create new configuration files for your instrument, but with fewer channels than the full configuration. This enables faster scanning.

IMPORTANT: If you have a previous PPM version installed on your computer, you must remove this version before installing a new version. First go to Settings→Control Panel→Add/Remove Programs. Then Remove PinPoint Monitor. If you get the message "Remove Shared File?", click Yes to All. And respond Yes you wish to proceed.

The software is provided in a self extracting installer: PPM Install.exe. Double click on this file to install the software on your computer. The install will place icons on the desktop for PPM, PPS, PPV, and PPP. It will also provide a shortcut to PPM Readme.txt, which is a file to get you started with the software.

HARDWARE FILES

Important files particular to your hardware are in a second self extracting file named: <your serial number> install.exe. These files must be installed after PPM is installed. To install these files just click on the “<your serial number> install.exe”. Click through Next, and Finish.

The hardware files will be installed to a folder with the same name as the Serial Number of your hardware. They will appear on your hard drive at C:\Program Files\GEC\PinPoint\<serial number>.

Should these files become corrupted while running PPM, they can be restored from the self-extracting hardware install file.

RUNNING THE SOFTWARE

The default Com Port is Com 1. Default temperature units are °C. Run PinPoint Settings if you wish to change either of these settings. Select your desired settings, click OK and then answer Yes to "Apply New Settings".

If your installation is complete and if your hardware is powered up and connected to COM1, (or to another Com Port selected in PPS) you should be able to start the PinPoint Monitor program and begin taking readings. The first time you run the program it will automatically ask you to create a new workspace name (*.ppw) and a new input monitor name (*.ppi). If no readings appear, go to [TROUBLESHOOTING PROCEDURES](#) on page 20.

When you start PPM, the program communicates with the instrument through the serial port and tells it to start sending data. The instrument will then start sending data continuously until you exit PPM.

The instrument operates in scanning mode. Once it is started, it sends the data from all channels on a continuous basis over the serial port whether or not you have a sensor connected to every channel. The scan interval is the update interval for your data.

While the program is running, the most recently sent data are displayed in the Real-Time Report, which is located at the left side of the screen upon program startup. These are the current temperatures and they are updated at the scan interval.

The Real-Time Graph is to the right of the Real-Time Report. This graph plots the temperatures, one or more pixels at a time. It starts plotting automatically as soon as the program is started. The number of pixels available for the plot depends on the size of your window and the resolution that you have set for the display. This plot is Optimized automatically on startup by scaling the time for the X- Axis so that each new data point in the plot is equivalent to one pixel.

THE MENU BAR

Many program features are controlled from the Menu Bar located at the top of the main window.

File: This menu provides the standard Windows file open and save features, as well as commands to perform on the active document and on the workspace. The bottom of this menu contains a recent file list, any of which can be opened by clicking on the file name.

View: This menu toggles which toolbars and windows should be displayed.

Tools: This menu gives access to additional tools, including Alarm Settings, the Calibration Wizard, Copy Real-Time Report, Configure Differential Thermocouples, and Program Startup Options.

Windows: This menu provides capability for arranging and sizing the various windows of the application.

Help: This menu accesses the built in Help system for PinPoint Monitor. You must have an IE browser on your computer to open the PPM Help file.

THE MAIN TOOLBAR

The Main Toolbar provides a convenient and visual way to access the functionality of the Menu items. As you move the mouse pointer over each icon, a descriptive panel will appear for that item.

New Workspace: The leftmost icon is used to create a new workspace, *.ppw. Each time the PPM program starts you must create a new workspace or load a previous workspace. A previous workspace will automatically be loaded on program startup if this option is selected. PPM can have only one workspace at

any given time. If you change the default temperature units within PinPoint Settings (PPS), then you must not try to load a previous workspace created with different units.

New Input Monitor: The next icon is used to create a new input monitor view, *.ppi. Please note that you may create multiple ppi views, each with its own selected channels and graph settings.

New Log Manager: Use this icon to create a log manager, *.ppl which will keep track of the details of your log files.

Toggle Workspace: Click this icon to open or close the Workspace. The Workspace is a main component of PinPoint Monitor. It manages views, logs, calibration files and hardware settings. Normally the workspace will be closed.

Toggle Device Status: Click this icon to open or close the Device Status window. This window reports the actions of the hardware components. Normally this window will be closed except when diagnosing a problem with the operation of the hardware. If the hardware is functioning properly, its operation will be transparent to the user.

Switch to Input Monitor: Click this icon to bring the input view (*.ppi) containing the real-time readings and the plot to the front. Multiple clicks will cycle through all input monitors (*.ppi) if more than one is loaded in the workspace.

Switch to Log Manager: Click this icon to bring the log manager to the front. Multiple clicks will cycle through all log managers if more than one is loaded in the workspace.

THE STATUS BAR

The Status Bar at the bottom of the screen indicates the connection port, the mode (normal or simulation), the scan interval, the configuration file (*.ppc), the calibration file (*.cal), the hardware file (*.pph), and it shows how many logs are active.

THE REAL-TIME REPORT

When the program starts up, the temperatures from all channels are displayed and updated at the scan interval in a dockable window at the left of the screen. This window may be moved to a different position if desired. Channels with sensors not plugged in or not working will display as OPEN. Note that the size of the windows in Pinpoint Monitor can be adjusted vertically or horizontally by moving the mouse pointer to the border of the window (a double edged arrow will pop up), and then dragging the border to make the window larger or smaller.

You may grab the right border of the Real-Time Report with the mouse pointer and move it to the right. This will reveal additional headings for Units and for Time Stamp. The Units column is used to display current units for each input. To change units for any channel, click anywhere on the line for that input. Then double click on the name of the unit. A box will open with the name of the available units. Click the desired unit, then click anywhere in the report to update your selection. Note that you can change the width of any column in the Real-Time Report by grabbing the interface between column headings with the mouse pointer and moving left or right.

The Options Dialog (Tools Menu → Options) allows you to configure the precision of thermocouple and thermistor readings that appear in the Real-Time Report. The defaults are two decimal places for thermocouple probes and four decimal places for thermistor probes. If you change these settings, they are saved to the registry and reloaded the next time you start the program. NOTE: These settings do not affect graphing or logging.

The Aliases

The labels under the Input heading are in the form of aliases. That is, the user can edit these labels to provide a more meaningful description of what that input represents. For example if a label reads TC1, this means thermocouple number 1. But if thermocouple number 1 is in the center of a seed, you may wish to change the TC1 label to read Center of Seed. Just click on that line, double click on the label, and when you get a blinking cursor you can edit the label. Then click anywhere in the report to update the label.

It is advantageous to keep the aliases to as few characters as possible because the Aliases you enter will subsequently appear elsewhere in the program such as graph labels, and channel labels for logging and exporting data.

Once you have edited the Aliases, you may wish to save this custom input view with File → Save All. The Aliases and the default units for each input are saved in the workspace file *.ppw. You are allowed to have more than one file with this extension, but only one *.ppw may be loaded at a time. If you wish to save different Views with different aliases or different units you should create a new workspace prior to editing the Aliases or editing the units, then save the workspace.

If you are going to edit the Aliases, you must do so before creating a log file. Once you have created a log file within a given workspace, you will not be able to edit the aliases while that log is active. If no logs are active, then you can edit or re-edit aliases.

THE REALTIME GRAPH

The right portion of the screen contains the Real-Time Graph. Channels to be plotted can be toggled on or off individually by moving the mouse pointer and clicking the box to the left of the channel ID. You may click on one or more boxes to turn those channels on or off as desired. Do not be concerned if the software does not respond immediately to your clicks on these boxes. Sometimes they will not be updated until the beginning of the new scan, i.e. you may have to wait all or part of a scan interval to see a response to your mouse clicks on these boxes.

The background of a box will be white whenever a channel is turned off. When it is turned on for plotting, the box will have a color representing the color of the graph for that channel. Clicking the box in the header toggles all channels on or off. At least one channel will be on at all times.

The elapsed time of the Real-Time Graph is displayed in a bar above the graph. This bar also includes the current time from the clock on your computer. Clicking Reset will restart the Real-Time Graph.

Y-Axis Settings is used to select the scale for the Y Axis.

X-Axis Settings provides a window to select X Axis plotting options for the Real-Time Graph. Click Optimize to automatically select a page size that will be scaled such that each new incoming data point will be plotted at the next pixel on the graph. The time units on the Real-Time Graph may be in days, hours, minutes, or

seconds. Please refer to the section [REALTIME GRAPH PLOTTING OPTIONS](#) for further information on plotting features.

Unit is used to select the units for the Y-Axis. If your system includes multiple types of units such as temperature and relative humidity, you can only select one type of units for a particular Real-Time Graph. If you desire to see Real-Time plots for more than one unit, simply create another input monitor (*.ppi) and select the desired units. Each time you select different units in a given *.ppi, the Real-Time plot will be restarted automatically.

DATA LOGGING

If you want permanent data that can't be lost accidentally, then you should start a log as soon as you start the program. Do not rely solely on the Real-Time Graph to save important data. Log all channels of importance at whatever interval is needed.

PinPoint Monitor logs selected channels to a database (*.ppd). Any or all channels may be selected for logging at user specified time intervals. Logged data may later be retrieved from the database by exporting to another file format such as Excel.

Before logging can begin for the first time, you must create a new log manager. Click on the third Icon in the Main Toolbar [New Log Manager]. Give the log manager file a name. It will have extension *.ppl. Click Ok and the log manager view will be displayed. This is basically an organizational table that will show the details of the log sessions that you have created.

To add a log session to the log manager, click on Edit → New Log. Enter a filename for the log and click on Save. A window will pop up to enable selecting which channels to the log as well as the log interval and the log mode. Select or deselect all channels by clicking the box at the top of the window.

The log mode can be Timed or Direct. Direct mode logs every reading of the hardware by automatically selecting a time interval equal to the scan interval. In Timed mode, select the desired time units for the log interval from the drop-down menu, and enter the desired time for the log interval, with or without a decimal point as desired.

The default units for logging will be those in the Real-Time Report. If you wish to log in different units, click on the unit name and select other units. These must be edited channel by channel.

Select 'Instantaneous' to log data from the selected channels at the time interval selected. You may also select parameters to log statistical data including average, standard deviation, maximum, minimum and spread.

When you have finished making your selections, click Ok.

Note that a new line is added to the log manager. The number of channels in the log appears in the second column. Note that the Time Interval and the Log File name are shown at the right hand side of the line. If you wish to add a description to the log, click under Description and edit <unnamed log>.

Click on the box at the left of the experiment line to manually start the log session. Or you may right click anywhere on the line to open a dialog that will enable you to select a date and time to start logging later. If you select a delayed start time that is prior to the current real time, the log will start as soon as you close this window. You may also select a date and time for the log to stop. The log session will continue as long as the program is running and the box at the left of the line is checked. You may toggle this log session on and off at any time with this check box.

Note that the time that the log was started is automatically displayed. Unless you have selected a time for the log to stop, the end time is listed as <...active...> whenever the log is running. If the logging is stopped, the end time of the log will be automatically displayed.

If you wish to later view which channels have been selected for logging in a particular session, highlight that session, and double click anywhere to the right of the checkbox. You may also use the "View Log Settings" option under the Tools menu to see details of a particular log after it has been created.

You may add other log sessions to the log manager as desired. These may contain the same or different channels and the same or different log intervals as previously created log sessions. And you may have several log sessions running simultaneously if desired. Note that it is not possible to have multiple log sessions using the same database (*.ppd) simultaneously. You should turn off log sessions before exiting PPM.

Note that you may wish to create new log managers (*.ppl) for different users or different projects.

RETRIEVING DATA FROM A PPM DATABASE

Data may be retrieved from a PPM database at any time by exporting to another file format for permanent storage. And logged data in a PPM database may be viewed or exported at any time by running the stand alone PinPoint Viewer program described on page 23 of this document.

To export data from within PPM, first open the log file window, *.ppl. Then select the line for the log session from which you wish to retrieve data. The log session is selected by clicking anywhere on the line. This log session will be highlighted. To export data from this log session to a permanent file, click on Tools → Export Log. This will open a window in which you will select the channels for which you wish to export data. The window's title bar will show the number of entries in this log. Click in the box at the top of the window to select or deselect all channels. If you do not wish to export all channels, click on those you want to export.

Exporting will retrieve every data point logged for the time period within the Start Time and End Time. Note that the beginning and end time of the log session are automatically inserted. If you do not wish to export all the data, you may edit the Start Time and or the End Time before exporting. If you wish to export all the data, simply make sure that the Start Time is less than or equal to the begin time of the log and that the End Time is equal to or greater than the end time of the log.

Note that you may edit the Description field if you wish, up to a maximum of 32 characters. Valid characters include only letters and numerals. Or you may leave the name of the Description field blank.

You may select the type of file to export to, including a spreadsheet file, HTML document, or text file.

Under Location, you must enter a filename to export the data to. The export will have a time stamp in the format Month/Date/Year HH:MM:SS. Further, you may select an additional column of elapsed time starting with 0 for the first data point. Units for the elapsed time may also be selected. And if you do not want all the readings, you may enter an integer number in the Filter. Once all selections have

been made, click Ok and the export will begin. An hourglass will appear during the export process.

Sometimes when exporting to Excel, you may get the message: Can't Update. Database or object is read-only. If you get this message, then usually it will work correctly if you try it a second time.

The message: Can't Update. Database or object is read-only sometimes occurs when the ppd file is saved in a path that is over 64 characters in length. It only appears intermittently. In order to avoid this error, try to keep path names to less than 64 characters. (For example, "C:\data\filename.ppd" instead of "C:\data\folder\second folder\third folder\fourth folder\fifth folder\filename.ppd")

Keep in mind that path names to My Documents or the Desktop are longer than "My Documents\File name" or "Desktop\File name" (The real path is to My Documents is "C:\Documents and Settings\\My Documents\File name.ppd.")

Note that the upon opening a standard Excel spreadsheet, the time stamp in the first column will appear as MM:DD:YY. The seconds will appear to be missing. This behavior is standard Excel behavior. The HH:MM:SS units are there, but in order for them to appear, first the column must be reformatted with Format → Cells → Number → Custom. Under Type, pick m/d/yyyy h:mm. Then add :ss and click Ok.

HARDWARE SETTINGS

You may edit hardware settings by opening the Workspace and double clicking on Hardware 0. This will open the Hardware Configuration dialog. You may edit the COM port, the baud, data averaging, filter (this is a low pass filter), and data rate. Choose a data rate of 60, 30, 15, 12 or 10 to reject 60 Hz noise (or 50, 25, 16.6667, 12.5 or 10 in areas with 50Hz power). Note that the available scan interval range (min. to max.) automatically updates as you change other parameters. For any combination of parameters, the default scan interval shown is the same as the minimum scan interval. If you wish to receive the readings from the instrument at a different interval, you can enter any desired scan interval (in units of seconds) within the available range shown. You may save these new settings under a new filename with a .pph extension. If you do not save the settings manually, they will automatically be saved in a file called "new_settings.pph" when you press OK to leave the Hardware Configuration dialog. Note this file will be added to the

workspace, which means it will be automatically reloaded whenever this workspace is loaded.

CALIBRATION PROCEDURE

The GEC Instruments system uses unique hardware and software procedures to enable very stable and highly accurate temperature measurements. The unit has been carefully calibrated prior to delivery. However, the user may wish to calibrate the entire system, including user-installed thermocouples, to provide the utmost accuracy available from the equipment. Please refer to the Hardware manual for your particular instrument for guidance on how to accomplish a good calibration. If your instrument has inputs in addition to temperature, such as relative humidity, you may use the calibration procedure within PPM to calibrate these inputs also. If you are calibrating an instrument with multiple types of inputs (such as temperature and humidity), be sure to select only one type of input at a time during a calibration.

The thermocouple calibration procedure involves generating a set of correction factors for each thermocouple channel and storing these in a calibration file, *.cal. If a one point calibration is performed, this file will contain a list of temperature offsets which will be applied to those channels which were calibrated. If a two point calibration is performed, the *.cal file will contain a list of slopes and offsets which will automatically be applied to those channels that were calibrated. If the user wishes extreme accuracy in a particular range, the system should be calibrated with a two point calibration at two temperatures near the ends of the range of interest.

A new calibration file *.cal is created and loaded into PinPoint Monitor whenever you calibrate thermocouples. If a mistake is made during calibration, you can hit the back button within the calibration Wizard. And you can always go back and recover a previous calibration.

To start a calibration from within PPM, click on Tools → Calibration Wizard. This will open a dialog (Calibration Wizard: Step One) where you can select a one point or a two point calibration. Or if you wish to load an existing calibration file, select Load User Calibration, and click on a filename under User Calibrations. Now select the temperature units of the reference device that you are using to calibrate against. These units do not have to be the same as the units you are using in PPM, as long as they are the same type of units. That is, you can input readings

in °F from a standard thermometer even if you are using °C within PPM. If you are calibrating humidity inputs, select units of percent.

If you are doing a new calibration, you may wish to select one of your existing thermocouples (or a precision thermistor or RTD probe if your unit has either of these) to calibrate other thermocouples against. In this instance, choose Use Reference and then click on the box below to select which input to use as a reference.

Next you must select the inputs to be calibrated by clicking on the associated boxes in the input list. Remember that the system may not respond to these clicks until the end of the scan interval. Click “Next” and go to Step Two: Gather Calibration Information. Place the probes together with a reference device into a steady environment. Follow the directions in the dialog. When readings become stable, enter the value from your reference device into the slot at the lower left. If you are using an existing thermocouple or other probe connected to your instrument as a Reference there is no need to enter a value, this probe will be read automatically.

Now click Acquire. If you make a mistake in entering the temperature or if the readings are not stabilized you can enter a new value and click Acquire again. Or if you have made wrong selections on the previous screen, you can click the Back button and make corrections. Once you are satisfied with your readings, click next to go to Calibration Point number 2 (this point will be skipped if you are doing a one point calibration).

Repeat the procedures for point two, then click next to go to Final Step: Calibration. Here you may view the calibration details. Since the details are selected in blue, you may copy them to the clipboard (Press Ctrl and C) so they may be imported into a text file for a permanent record.

If you are satisfied with the calibration, press Finish. The calibration corrections will automatically be applied to your inputs.

Now enter a file name for your new calibration file. You may click on the browse button at the right to negotiate a new path if desired. Once you have entered the file name, click Ok and the calibration file will automatically be loaded into the workspace. The calibration file name will appear in the Status Bar as well.

You may have only one calibration file loaded at a time. If you wish to go back to the default calibration, open the workspace and highlight the calibration file *.cal.

Press delete and the *.cal will be removed from the workspace and the default calibration will be restored.

Be careful when loading previous calibration files, particularly if you are restoring calibration constants for only some of the inputs and not all of them.

ALARMS

Alarms allow each input probe to be monitored and an arbitrary action can be triggered whenever some threshold is exceeded. Choosing Alarm Settings from the Tools menu displays a dialog that gives access to the parameters defining an alarm and the action to be taken when it is triggered. Each channel may have a high alarm and a low alarm associated with it. A high alarm activates whenever the temperature equals or exceeds the high alarm threshold. A low alarm activates whenever the associated temperature equals or goes below the low alarm threshold. Alarms may fire once only (the first time the condition is satisfied), or whenever the condition is true.

An alarm action may be one of "System Beep", "Play Sound", or "Execute Command". System Beep & Play Sound are similar; the difference is that Play Sound plays an arbitrary .wav file of your choice. Execute Command is a powerful option, which lets you construct an arbitrary command line using the parameters of the alarm event. Here are some examples:

```
cmd /c echo Alarm fired >> C:\AlarmLog.txt
```

This will append the line "Alarm fired" to the file C:\AlarmLog.txt, whenever the alarm is triggered. To make available more information about the alarm event, you may include special %-format specifiers on the command line. Valid substitutions are:

%%	-	% character
%t	-	timestamp
%a	-	current alias of offending probe
%v	-	value of offending probe in alarm threshold's units
%r	-	alarm threshold (including units)
%y	-	alarm type (Hi or Low)

So, if instead the command were written

```
cmd /c echo Alarm fired: input %a exceeded threshold %r at %t >>  
C:\AlarmLog.txt
```

then, every time the alarm is triggered, a line of the form

```
"Alarm fired: input "REF" exceeded threshold 50 Celsius at 15:23:46 13 February,  
2004"
```

will be written to the end of the file c:\AlarmLog.txt.

CONFIGURING DIFFERENTIAL THERMOCOUPLE INPUTS

Differential thermocouple inputs can be used to read small temperature differences to very high accuracy. Such inputs read the difference in temperature between two thermocouples in a circuit where one thermocouple is the reference junction and the other is the measuring junction. For best accuracy, the approximate temperature of the cooler side (reference junction) must be known. Choose Config Differential TC's from the Tools menu to display a dialog that enables the user to set the temperature of the cooler side to a fixed value, or to select another probe as a reference for the cooler temperature. A separate cool side temperature can be set for each differential thermocouple input. If these values are not set, they are 0 °C as a default.

CLOSING THE PROGRAM

You may exit PinPoint Monitor with File → Exit or by clicking on the X at the upper right of the main window. If any log sessions are active when you attempt to exit, you will get a message to this effect. You must close these logs to continue exiting.

THE PPM DYNAMIC DATA EXCHANGE SERVER - DDE

Dynamic Data Exchange (DDE) is a process whereby data can be shared among multiple Windows programs (including Labview) running on the same computer or over a network. PinPoint Monitor implements DDE with the DDEML.

Additional information on this is available online through MSDN. For the DDE Server to respond, PinPoint Monitor must be running with a *.ppi loaded (i.e. you must see a Real-Time Report and Real-Time Graph running).

Service Name

Pinpoint registers under the default service name "PPMonitor". This can be changed by the user by adding a file called "DDEConfigFile.ppn", containing the desired DDE server name and nothing else, to the directory containing the *.ppc file (the configuration file for your instrument). The ability to register under different service names is useful when running multiple instances of PPM on the same computer or the same network.

Topic Names

Pinpoint currently supports two topics: "System" and a topic named after the *.ppc file that is loaded. For example, if you are running AH01.ppc, then there will be a topic "AH01.ppc". If you would like to determine this programmatically at runtime, you can request the item "Topics" from the "System" topic, and it will return the correct *.ppc name.

Items

Under the relevant .ppc topic, the following items are supported for advise-loop notification:

- | | |
|--------------------------|--|
| "Average Scan Interval", | the scan interval, in seconds |
| "Probe Names", | a comma-separated list of probe names |
| "Scan Start Time", | a timestamp for the beginning of each scan |
| "Temperatures", | a comma-separated list of current temperatures |
| "Temperature Units" | the current temperature units |

REALTIME GRAPH PLOTTING OPTIONS

The Real-Time Graph has many optional features that may be set by the user. Click on the X-axis Settings to edit the time settings for the Real-Time Graph. The Duration is the total time on the x-axis of the Real-Time Graph. When the plotted data has reached the end of the graph, values will continue to be updated at the rightmost side of the graph, and previous values will move to the left at each update.

The Duration is the time period for which data will be stored in memory. For example, if the duration is 300 minutes, then the program will store and plot data for 300 minutes at the update interval shown. The update interval for the Real-Time Graph is not necessarily the same as the scan interval. Rather it depends on the number of pixels available in the plot, graph duration, etc. If the Update Interval indicates "Hardware Limit", then the graph will be updating as frequently as possible, at the scan interval, which is the same rate as the new data are coming from the instrument.

A particular chart may be divided into several pages by enabling the scrolling option and entering the time for each page. Note the update interval shown. If you click Optimize, the Real-Time Graph will be set up to plot with a density of N readings per pixel, where N is "Scans per pixel" in the dialog, whenever the amount of time indicated by the update interval has elapsed. For this reason, there is a fixed relation between duration, time length per page and update interval. And this relation depends on the resolution at which the monitor is operating. The user cannot set the update interval, however, the user can modify the hardware scan interval which directly influences the update interval. The update interval is changed automatically based on the selections made for units, duration, and page length.

The points being plotted are being saved in memory and the program allocates about half of the available memory to the Real-Time Graph. If a very long duration is selected, then the chart will be limited by available memory and the update interval will be increased accordingly. If multiple input monitors (*.ppi's) have been created, the available memory is automatically shared among the multiple Real-Time graphs.

If the update interval is at the hardware limit, this means that points on the Real-Time Graph are being plotted as frequently as possible, which is at the same rate as they are being acquired by the hardware. In this instance the chart may not take up

the full screen and there may be unused white space to the right of the graph. If this occurs, increasing the Page value will widen the graph and allow more pixels to be plotted at the same interval. Click Optimize to do this automatically.

If "Hardware Limit" is grayed out, then the graph will update at an interval that is more than the scan interval. But remember that the Real-Time Report continues to update at the scan interval.

As an example, if you set the graph duration to 300 minutes, then after 300 minutes the new data will continue to be plotted, but the plot will start scrolling to the left. That is, the current data will always be at the rightmost position of the graph. Each time a new point comes on at the right, the graph shifts to the left by one pixel and the leftmost point is deleted. In this way the last 300 minutes are always displayed. So after the graph reaches the 300-minute mark, it will plot the current data at the rightmost part of the graph but the Elapsed Time clock will continue past 300 minutes.

The X-Axis Settings has a Scrolling option. If Scrolling is not checked, then the Real-Time Graph will be one page. If scrolling is enabled, you will have multiple pages, and you can select the minutes per page.

If scrolling is enabled, a scrollbar will appear at the bottom of the graph. Only one page will be displayed at a time. The total length of the graph will still be the same as the Duration. Once the plot has reached the rightmost position you will need to use the scrollbar at the bottom to move the graph to the left so that you can see the latest data. In other words, the current value for Elapsed Time must appear on the X-axis in order to see the latest point being plotted.

The advantage of using multiple pages (scrolling) is that you can look at current data or you can easily scroll back to look at previously plotted data. The number of data points that can be stored depends on the amount of memory available on your computer. Most modern computers can save many hours of data in the Real-Time Graph.

When using X-Axis Settings, certain combinations of settings are incompatible and may result in an unusual looking graph or graph scale. If this occurs, simply do X-Axis Settings again and change settings for update interval or for some other unit.

Occasionally the chart may become scrunched to the left with all of the time axis values located under the vertical Y-axis. If this occurs, simply grab the leftmost

side of the chart area with the mouse pointer and move it one pixel to the right or left. This action should expand the chart to a normal view.

Each channel may be plotted in a different color on the graph. There are several ways to change the color of a channel. You may select or change graph colors with the Color feature described below, or with the Properties feature, or by opening the Palette Bar under Tools and dragging a color to the line that you wish to change.

Another legend identifying the color for each input may be selected by right clicking within the graph area and selecting Legend Box. The legend box features may be edited by right clicking in the legend box. These features include Hide, Autosize, Font, Fixed, Floating, and position relative to the chart – left, top, right, or bottom.

Touching the mouse pointer to any line being plotted will pop up a box with the channel ID and the temperature reading. This is a very useful feature to quickly identify which plot is which. If you wish to change the color of this line, simply right click when you see the pop up box and select a new color by clicking on one of those colors displayed.

You can easily change the background color of the graph or of the graph border by positioning the mouse pointer in the desired area and right clicking to open a view of the palette, then click on a new color to select.

A toolbar at the top of the graph provides the following functions:

Save Chart -- this saves the current chart (you must give it a filename and you may select file type including ChartFx files, Metafiles and Bitmap files.)

Copy to Clipboard -- copies the current chart to the clipboard

Gallery -- used to select the style of the chart. The line chart is most useful.

Color -- The default color is shown in the rectangular box at the bottom of this icon. To change the default color, click on the icon and select a different color. This icon is used to change the color of the graph outline, the graph background or to change the color of an individual line being plotted. To use this feature, click on the icon with the left mouse button and hold down on the button. Drag the pointer

of the paintbrush to the item that you wish to change color of and release the mouse button.

You may also change the color of a line by positioning the mouse pointer on the line, then right click to open a window where you may select a different color.

Vertical Grid -- toggles the vertical grid

Horizontal Grid -- toggles the horizontal grid

Data Viewer -- clicking on this icon opens or closes the Data Viewer. The Data Viewer is a very useful feature. It lists the temperature values for each plotted point. Note that the data editor has scroll bars at the right and bottom sides of the window for the purpose of moving about within the Data Viewer. Note also that the border between the Data Viewer and the Real-Time Graph is moveable vertically. To move this border, position the mouse pointer at this interface line (a double ended arrow will pop up) and then click and drag the line to change the size of the Data Viewer window.

Properties -- opens a series of windows for adjusting properties of the Real-Time Graph. Note that each window has a help feature with further information applicable to that window by clicking on Help at the lower right. Whenever any graph property is changed, it is saved as a default and it will stay that way until it is changed again.

The General tab can be used to select the color for the background of the graph and the color for the border surrounding the graph window. A graph title may also be inserted or edited from this window.

The Series tab may be used to select line style, color and weight. Also markers for the plotted points may be selected.

The Axes tab may be used to select properties of the graph axes. The minimum and maximum values for the Y-axis may be selected with details. The values for the X-axis are not enabled in this window; rather they are selected with Adjust Chart.

Note: The ChartFx Properties dialog may be opened by positioning the mouse pointer in any region of the graph screen and double clicking with the left mouse button.

3D/2D -- toggles between 3D and 2D mode

Rotate -- not functional

Z Clustered -- not functional

Zoom -- this is a very useful feature. Zoom can be turned on or off. When turned on, the feature is used to quickly and easily change the scaling of the X and Y axes. The mouse pointer is used to select a rectangular area within some portion of the body of the graph. The portion of the graph within this rectangle will then be redrawn full size. First, the mouse pointer is positioned at the upper left corner of the desired rectangle and the left mouse button is pressed down to select the starting point for the rectangle. Then the mouse is moved to create the rectangle, which will enclose the portion of the graph to be redrawn when the mouse button is released. Note that a zoomed graph can be "rezoomed" over and over to keep changing the scale. However the graph can only be zoomed in. To zoom back out to the original "unzoomed" state, simply click again on the Zoom icon of the toolbar. Note also that a zoomed graph has a scroll bar at the bottom for moving the graph horizontally, and a scrollbar at the right for moving the graph vertically.

Print Preview -- this feature is self-explanatory

Print -- clicking on the printer icon will open the Print setup dialog. This allows the user to select the printer, the printer preferences and the page selection. Note that the "Pages" section of the Page Range defaults to "1-65535." This does not mean that there are 65,535 pages – it is standard behavior of the Windows XP print dialog.

Tools -- this icon is at the rightmost end of the toolbar. It is used to toggle the Graph Legend, the Data Editor, the Toolbar, and the Palette Bar (used to select the background colors or change the colors of individual lines on the graph. Just click on a color square and drag it to the graph body or the graph border or a line on the graph).

Note that whenever the mouse pointer is positioned within the body of the graph, a right click of the mouse button will pop up a box which will enable toggling the Toolbar, the Data Editor, or the Legend Box. Also selections may be made for adding or Editing a graph title, Font, and Chart Properties.

Note also that any border of the graph may be repositioned by grabbing the border with a left click of the mouse pointer (a double ended arrow will appear) and repositioning, then releasing the mouse button. Normally the right side of the graph window is the only one that may ever need to be repositioned.

TROUBLESHOOTING PROCEDURES

If your hardware is connected to the proper COM Port and you are not getting readings, go to the Device Status Window with View → Device Status. If errors are reported here, check the serial cable, make sure the hardware has power (red LED is ON) and make sure the instrument has signed on, (green LED is ON). The serial port on the computer must have DTR high and RTS low.

If COM1 is not working or not available, you may select a different COM Port in PinPoint Settings. And you can try operating at a slower baud rate. Open the Workspace and double click on Hardware 0. Select the desired Baud Rate and click OK. If a Workspace is not loaded, click File → Open → and select a *.ppw file and open it. If there are no *.ppw files you may create one with File → New → Workspace.

Note: The important files for your particular instrument are in the folder with the same name as the serial number of your instrument. They will appear on your hard drive at C:\Program Files\GEC\PinPoint\<<your serial number>. Should these files become corrupted while running PPM, they can be restored by running the hardware install.

The *.ppc contains the configuration and other necessary information for your particular instrument. This is the only file that is needed to run PPM.

The *.ppw file contains the aliases for the inputs, the default units for each input, and a list of files to be loaded on program startup.

The *.ppi file contains the configuration of the main input screen, which channels are selected for plotting, and the scale, units, update interval, etc for the Real-Time Graph.

The *.ppi.cht file saves any custom settings you have created for the Real-time Graph such as background colors, line style and width, etc. It will be accessed automatically whenever an existing *.ppi is loaded.

If you are having problems getting the program to run correctly, make sure the correct *.ppc is in the correct path. To check or change this, do Application → Options, or run PinPoint Settings. If you change the configuration file or the path you must leave the program and restart.

Note that the Tools → Options window has a box for Workspace Reload on Startup. The Workspace contains the channel aliases and the default units. The Workspace is normally reloaded on startup unless this box becomes unchecked due to a program crash. If this happens, simply check the box, click Ok, leave the program, and restart PPM. Or run PinPoint Settings and click Load Last Workspace.

If there are no files listed in the Workspace view, you can load a *.ppw file with File → Open. If you wish to create a Real-Time Report with different aliases or different default units, simply edit the aliases and/or the units and run Document → Save Workspace. Give the edited view a unique name. It will be a Workspace file, *.ppw.

If you wish to start PPM from scratch with only a *.ppc then do the following steps:

1. Close the Workspace with File → Close Workspace
2. Designate the configuration file (*.ppc) with Application → Options or with PinPoint Settings.
3. Create a new Workspace (*.ppw) with the leftmost icon on the main toolbar, or with File → New → Workspace.
4. Create a new Input file (*.ppi) with the second icon on the main toolbar or with File → New → Input Monitor

PLOTTING FROM A PPM DATABASE WITH PINPOINT VIEWER

Pinpoint Viewer is a standalone program which can be used to look at plots from *.ppd log files created by Pinpoint Monitor. The PinPoint Viewer can be utilized whether or not you are running PPM. It is used to plot data from previously logged databases or even a database that is currently being logged to by PPM. And multiple copies of the PinPoint Viewer may be started simultaneously, in the event you wish to plot from more than one database without closing existing plots.

PinPoint Viewer supports *.ppd log files with instantaneous readings as well as those that include values for average, standard deviation, max, min, or spread.

To use PinPoint Viewer, start the program, then click File → Open. Then double click on the desired filename. Alternatively you may click on a recent file if that is listed. This will pop up a dialog window that lists the channels that are in that database. Click those channels that you wish to plot. Adjust the X-axis and Y-axis units if desired. The start and end times are those that are at the beginning and end of the logged database. Adjust these times if you do not want to plot the entire log. If your PPM database (*.ppd) contains readings with more than one type of units (such as temperature and relative humidity), you must select the units you wish to plot. PPV can only plot one set of units on the Y-Axis at a given time. If you wish to view data with more than one type of units, you may run multiple versions of PPV simultaneously.

Click Ok to plot. Then click the boxes at the left to turn on or off the individual channels. Touch the mouse pointer to a plotted line for more information. Right click on any line to change its color.

Note that the filename and date and start time and end time of the plot are shown above the graph. Double click within the chart area to edit ChartFx Properties. Choose GEC Palette if available.

Click on the Data Viewer in the toolbar above the chart. This will display a list of the times and temperature values.

All of the chart features in PinPoint Monitor (except for X-Axis Settings) are available in PinPoint Viewer.

If you have more time values than horizontal pixels, you will not see every data point in the log, although all points will be loaded in memory. To look at more points in a particular region of the graph, use the zoom feature from the toolbar above the graph. Click the magnifying glass to turn on the zoom feature. Then position the mouse pointer in the upper left of the region of the graph that you wish to zoom. Hold down the left mouse button and select the region you wish to zoom by moving the pointer to the lower right of this region. Release the mouse button and the zoomed plot appears. Note that you will have a scrollbar for both the horizontal and vertical axis. This will let you look at other parts of the graph with the zoomed scaling. Click the magnifying glass to turn off zoom and go back to the original graph.

To create another plot from the same or another database, just run File → Open and double click on a database filename (*.ppd). If you have zoom enabled and then load another database, the axis scaling from the previous plot will be there. In this instance, turn off the zoom to see all of the data for the new plot.

And you may use Pinpoint Viewer to Export data to other formats such as Excel spreadsheets or text files. Just run File → Export and open the ppd file from which you wish to export. This will open a window in which you will select the channels for which you wish to export data. The window's title bar will show the number of entries for this particular log. Click in the box at the top of the window to select or deselect all channels. If you do not wish to export all channels, click on those you want to export.

The export will retrieve every data point logged for the time period within the Start Time and End Time. Note that the beginning and end time of the log session are automatically inserted. If you do not wish to export all the data, you may edit the Start Time and or the End Time before exporting. If you wish to export all the data, simply make sure that the Start Time is less than or equal to the begin time of the log and that the End Time is equal to or greater than the end time of the log.

Note that you may edit the Data Table name if you wish, up to a maximum of 32 characters. Valid characters include only letters and numerals. Or you may leave the name of the Data Table blank.

You may select the type of file to export to, including a spreadsheet file, HTML document, or text file.

Under Path, you must enter a filename to export the data to. The export will have a time stamp in the format Month/Date/Year HH:MM:SS. Further, you may select an additional column of elapsed time starting with 0 for the first data point. Units for the elapsed time may also be selected. And if you do not want all the readings, you may enter an integer number in the Filter. Once all selections have been made, click Ok and the export will begin. An hourglass will appear during the export process.

Sometimes when exporting to Excel, you may get the message: Can't Update. Database or object is read-only. If you get this message, then usually it will work correctly if you try it a second time.

The message: Can't Update. Database or object is read-only sometimes occurs when the ppd file is saved in a path that is over 64 characters in length. It only appears intermittently. In order to avoid this error, try to keep path names to less than 64 characters. (For example, "C:\data\filename.ppd" instead of "C:\data\folder\second folder\third folder\fourth folder\fifth folder\filename.ppd")

Keep in mind that path names to My Documents or the Desktop are longer than "My Documents\File name" or "Desktop\File name". The real path is "C:\Documents and Settings\\My Documents\File name.ppd."

Note that the upon opening a standard Excel spreadsheet, the time stamp in the first column will appear as MM:DD:YY. The seconds will appear to be missing. This behavior is standard Excel behavior. The HH:MM:SS units are there, but in order for them to appear, first the column must be reformatted with Format→Cells→Number→Custom. Under Type, pick m/d/yyyy h:mm. Then add :ss and click Ok.

USING CUSTOM THERMOCOUPLE COEFFICIENTS WITH TCCONFIG

Now users of GEC precision thermocouple scanners can easily input custom coefficients for calibrated thermocouple wire into a configuration file for their instrument by means of a new software application called TCConfig.

When using custom calibrations, the measurement accuracy of the system will be limited by the accuracy of the wire calibration and the homogeneity of the thermocouple wire. All other errors in GEC thermocouple scanners are minimal.

Download an installation of TCConfig with instructions from the downloads page at <http://www.gecinstruments.com>