



CHART INDUSTRIES, INC.

SCADA SOFTWARE USER'S MANUAL

*FOR SCADA 3.0, 3.1 & 3.2 (REAL CHAMBER MODELS)*

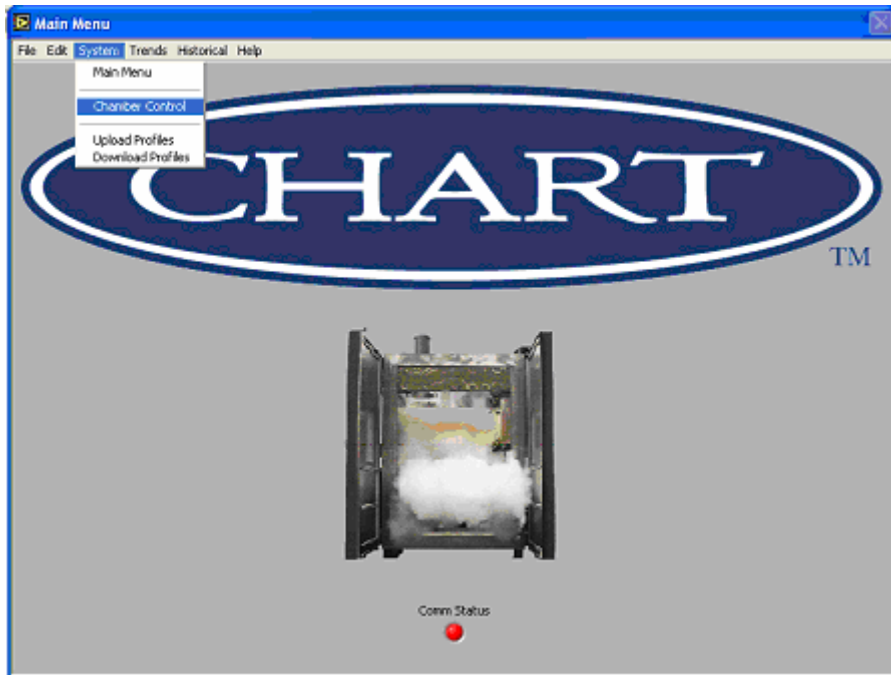


# Table of Contents

Chamber Control.....	3
<b>Selecting a chamber:</b> .....	4
<b>Chamber Configuration:</b> .....	4
<b>Viewing data:</b> .....	5
<b>Manual Mode Operation:</b> .....	7
<b>Automatic mode operation:</b> .....	8
Downloading Profiles (from PC to PLC).....	10
Uploading Profiles (from PLC to PC) .....	13
Creating Custom Trend Charts.....	16
The Custom Trend Screen .....	18
<b>Changing the Trend Chart Background or Grid Color</b> .....	19
<b>Changing the Trace Line Properties</b> .....	19
<b>Changing Custom Trend Chart Title</b> .....	20
<b>Changing Custom Trend Chart Scales</b> .....	20
Creating/Editing Profiles .....	22
<b>Editing an Existing Profile</b> .....	26
Profile Validation .....	26
Collecting Data .....	27
Reviewing Data.....	30
Exporting Data.....	33

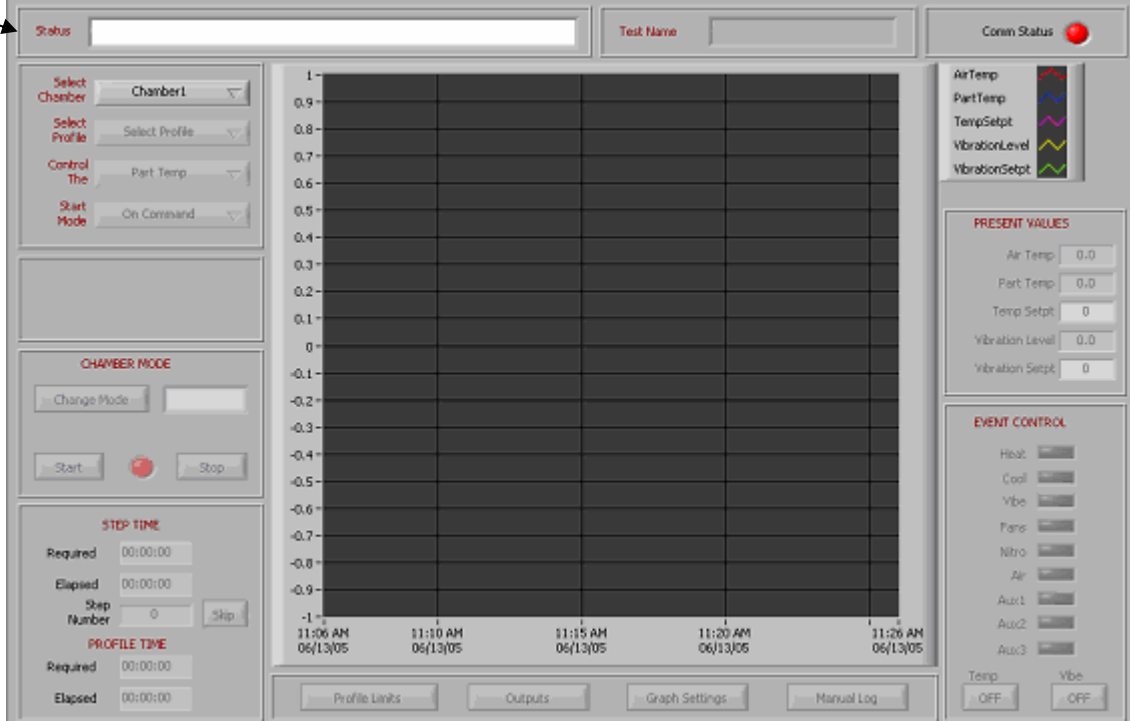
## Chamber Control

From the **System** menu, select **Chamber Control**.



Once the chamber control panel opens, ensure that there are no errors listed in the status box.

Status box

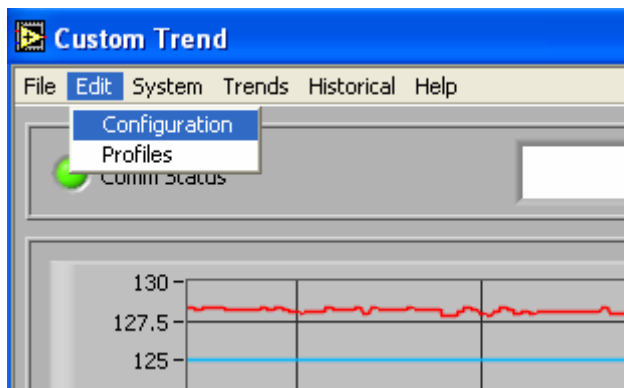


### **Selecting a chamber:**

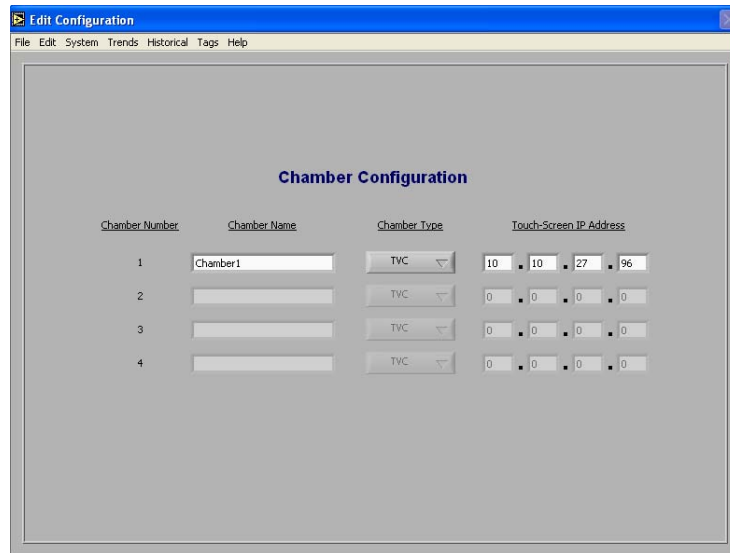
The box titled “Select Chamber” allows you to switch between multiple chambers that are connected to the network.

### **Chamber Configuration:**

To configure all chambers connected to the network select “Configuration” from the “Edit” menu.



A window will appear that shows the current configuration of networked chambers.

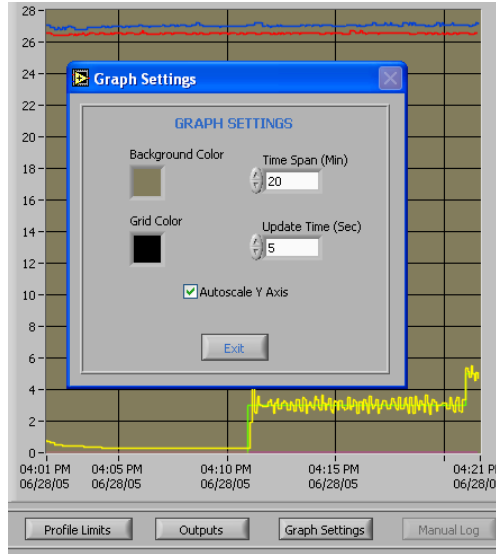


This window allows modification of the assigned chamber name, the IP address of the controller, and selection of the chamber model (vibration or thermal only). The IP address seen here needs to match the address setup in the CTI controller for proper operation. See the SCADA install manual for instructions on how to set the CTI controller address.

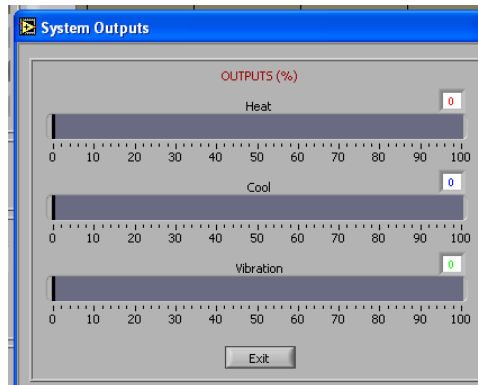
### **Viewing data:**

The graph in the center shows current chamber data. If the graph is blank, check the “Comm Status” light in the upper right corner. If this light is red, check the serial connection from the chamber to the PC.

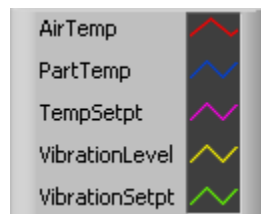
The graph settings change be changed by clicking the “Graph settings” button located below the graph itself. The time span setting controls how many minutes worth of data are shown at any one time (x-axis range). The Update time changes the number of seconds between graph updates. Chart recommends a value of 5 seconds to ease computer and communication resources. Both of these fields can be set by clicking the up/down arrow next to the value, or by clicking on the value and manually entering a number. You can set the background color and grid color by clicking on the box showing the current color and selecting a new one.



Just to the left of the “Graph settings” button is the “Outputs” button. This button brings up a window that will show the current heating, cooling, and vibration output levels.



There is a legend located to the right of the graph. Clicking on any legend entry opens a menu that allows the user to change color, axis, plot, and line options for that entry.



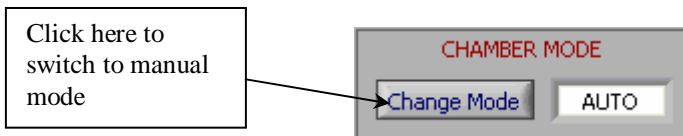
Below the legend is a box labeled test name. When a profile is being run this box shows the name of the file where data is being logged. Data logged from the chamber is normally found in the “historical data” folder. The file name and

data logging settings can be changed in the “historical setup” menu. See “Collecting Data” section below for further details.



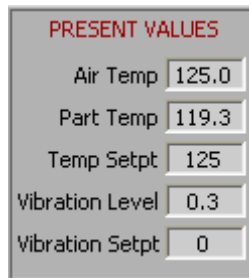
### **Manual Mode Operation:**

On the control screen there is a box labeled “Chamber Mode” which shows the current chamber mode and has a button labeled “Change Mode”. If the box shows “AUTO”, click the change mode button to go into manual mode. The manual mode works exactly the same as it does on the touch screen. It allows you to turn on/off events and change set points.

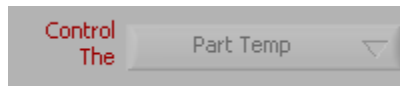


### **Changing set points:**

The set point values are located in the “Present values” box located to the right of the graph. This box shows the current chamber values and has two boxes labeled “Temp Setpt” and “Vibration Setpt”. To change the value, click inside the corresponding box and type in the new value.

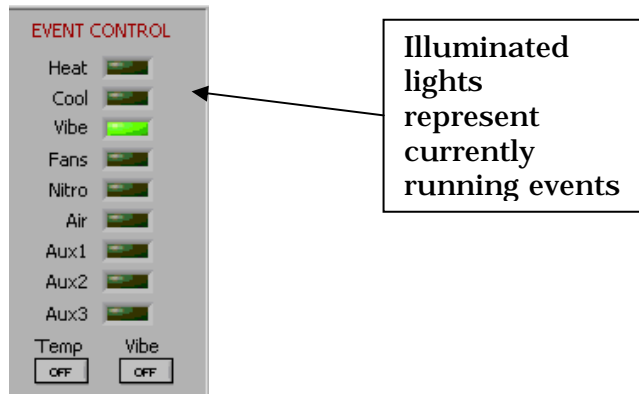


To the left of the graph is a drop down box labeled “Control the”. The drop down box has either “part temp” or “air temp” to choose from. To change between part temp and air temp click on the box and select desired option. This determines which thermocouple (part or air) the chamber controls the set point to.



### **Controlling events:**

The “Event control” box located in the lower right corner of the screen allows you to turn on/off chamber events.

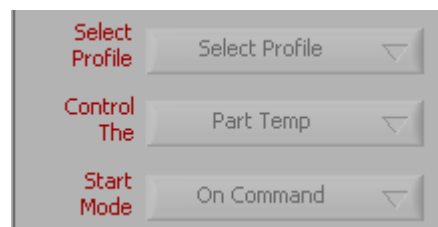


The top three buttons in this box control the heating, cooling, and vibration events. To turn an event on, first you need to click the green box next to the event name. Once the green button is illuminated the On/off box at the bottom will become active for that event. To start the chamber controlling to the temp/vibe set point found in the “Present values” box, click the “Off” button to toggle the event on.

The bottom portion of the event control box has buttons to control fans, nitrogen purge, air purge, and auxiliary events. These events are turned on/off by clicking the green box next to the event name. When the event is on, the green box will be illuminated.

### **Automatic mode operation:**

With the chamber mode set to “AUTO” the profile options to the left of the graph become active.



### **Running a profile:**

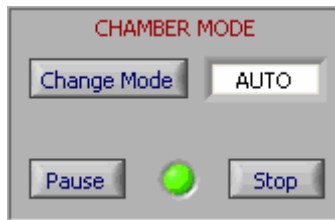
Clicking the “Select profile” box shows the current list of profiles that can be run. Once a profile is selected it will show up in the “Current profile” box to the left of the graph.

The “Start mode” box shows options for starting the profile.

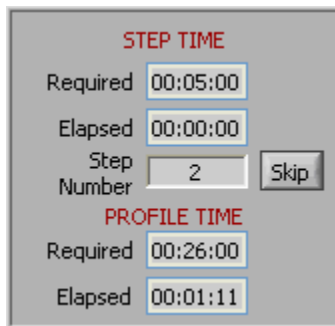
- On command: starts the profile when the user presses the start button

- Delay start: allows the user to delay the start of the profile. When this option is selected boxes open up below to allow the user to enter the desired delay time.
- Remote start: allows the user to supply an input that triggers the start of the profile (see CTI manual for further reference)
- At specified time: allows the user to enter the time which the profile will begin. When this option is selected boxes open up below to allow the user to enter the desired start time/date.

Once the desired start mode is set and a profile is selected, the Start button in the “chamber mode” box will become active. To start the profile, click the start button. To stop a profile during operation, click the stop button.



The lower left corner of the screen shows profile time data.



- Step time required: shows the length of the current step
- Step time elapsed: shows how long the current step has been running
- Step number: shows which step of the profile is currently running
- Step skip: allows skipping of the current step
- Profile time required: shows the length of the entire profile
- Profile time elapsed: shows how long the current profile has been running

#### Profile limits:

The profile limits box allows the user to control maximum and minimum limits during the profile to ensure safe operation.



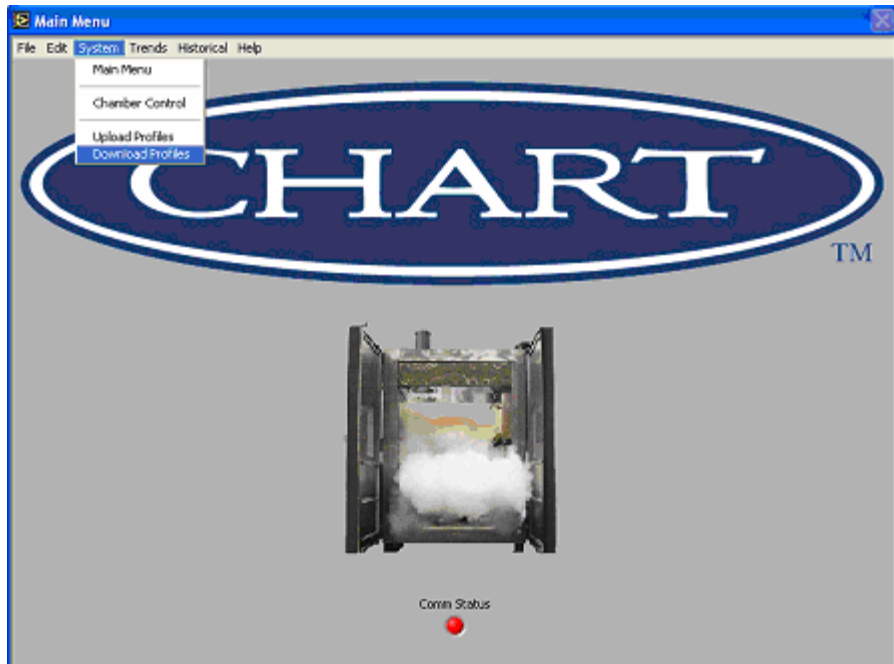
Clicking on the numeric value next to each limit allows the user to type in a value for that limit. The dropdown box next to the value determines what the chamber will do if the limit is reached.

- **Disable Output:** this option turns off the event which caused the alarm (example: if air temp hi is set to 100C and the temperature reaches 101C, the chamber will turn off the heating event)
- **Do Nothing:** the profile continues without interruption
- **Go to 20C:** the chamber controls the temperature to 20C and stops
- **Shutdown Chamber:** the chamber shuts down

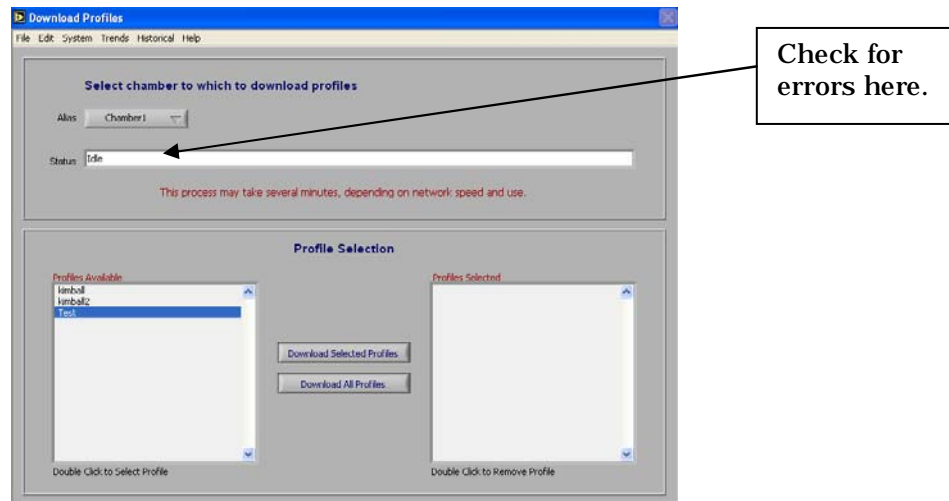
## **Downloading Profiles (from PC to PLC)**

The downloading utility is used to transfer pre-defined test profiles from the PC to the PLC. These profiles must be saved on the PC in order to download them into the PLC. There must also be an active Ethernet connection between the PC and the PLC (standard part of the SCADA package). The following procedure should be used to download profiles from the PC to the PLC:

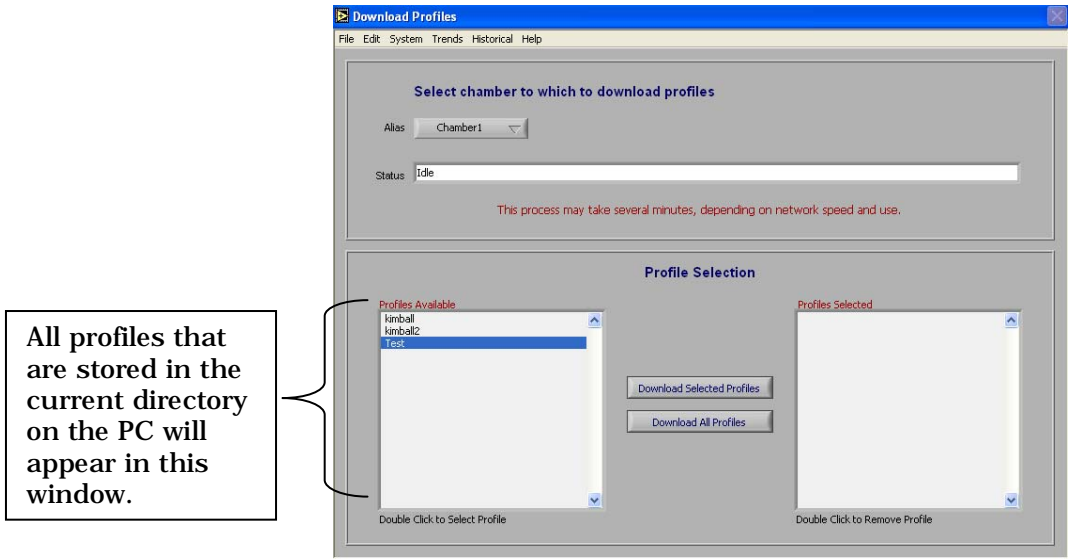
1. From the **System** menu, select **Download Profiles**.



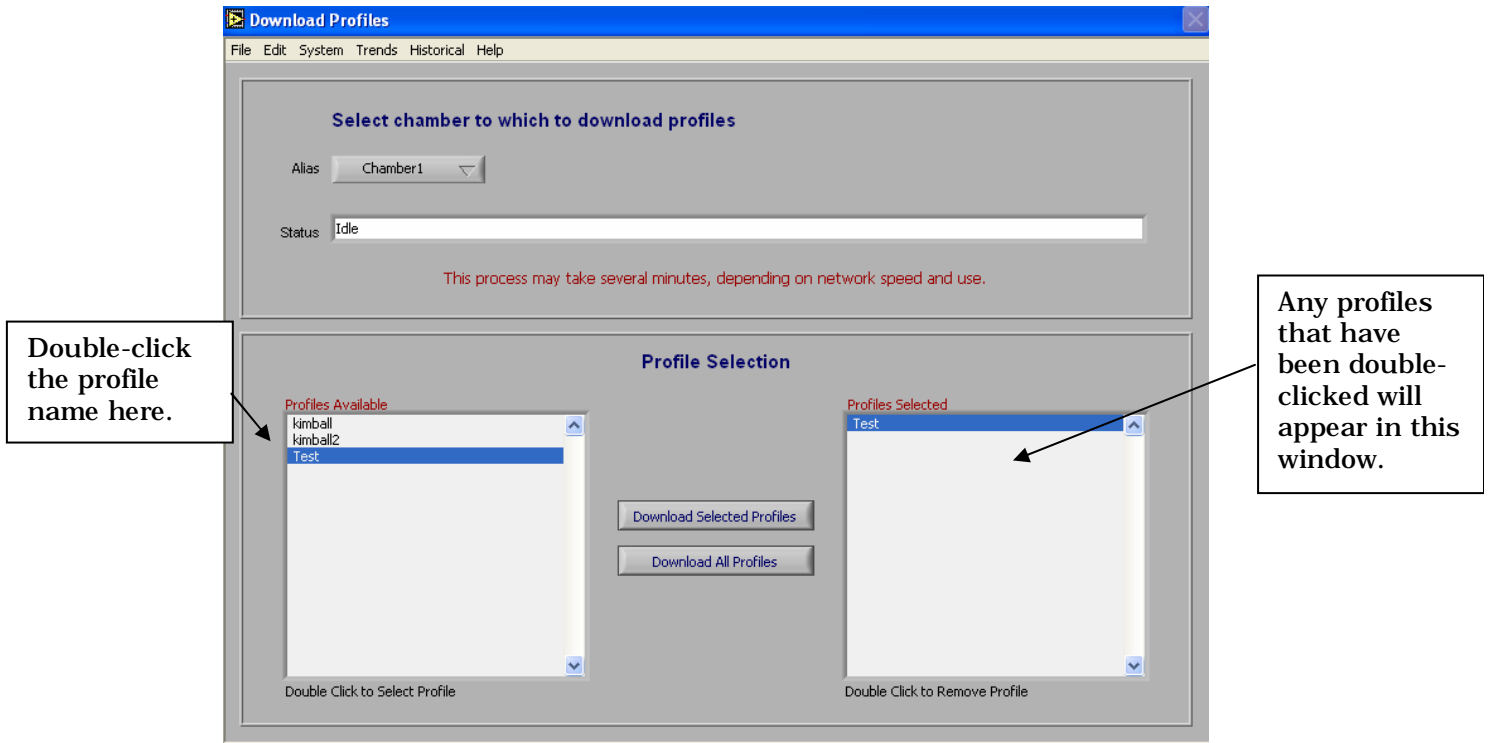
2. Verify there are not any error messages in the “Status” window.



3. The screen will show a list of profile names that are currently saved in the current directory. If the profile name of interest is not shown in the “Available Profiles” window, then it is not saved in the current directory.

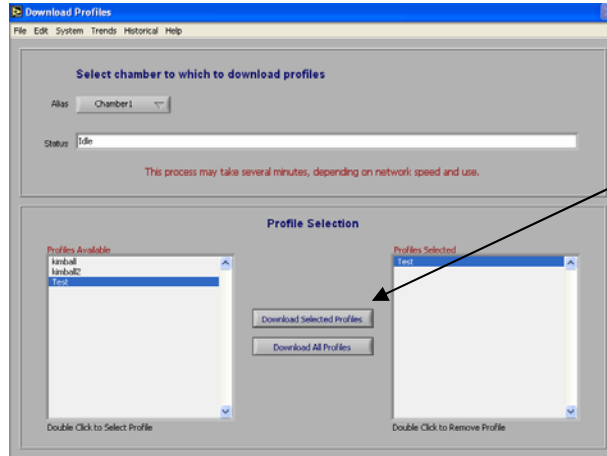


4. Identify the profile that is to be downloaded to the PLC.
5. Select the profile by double-clicking on its name.
  - Notice that when the profile is selected (by double-clicking its name), the name will appear in the “Profile Selected” window on the right.



6. Continue selecting profiles by double-clicking names until all profiles that are to be transferred appear in the “Profiles Selected” window.

7. If a profile is accidentally selected, it can be removed from the list by double-clicking the name in the “Profiles Selected” window, just the opposite of how it was added.
8. When all profiles are selected, click the “Download Selected Profiles” button.

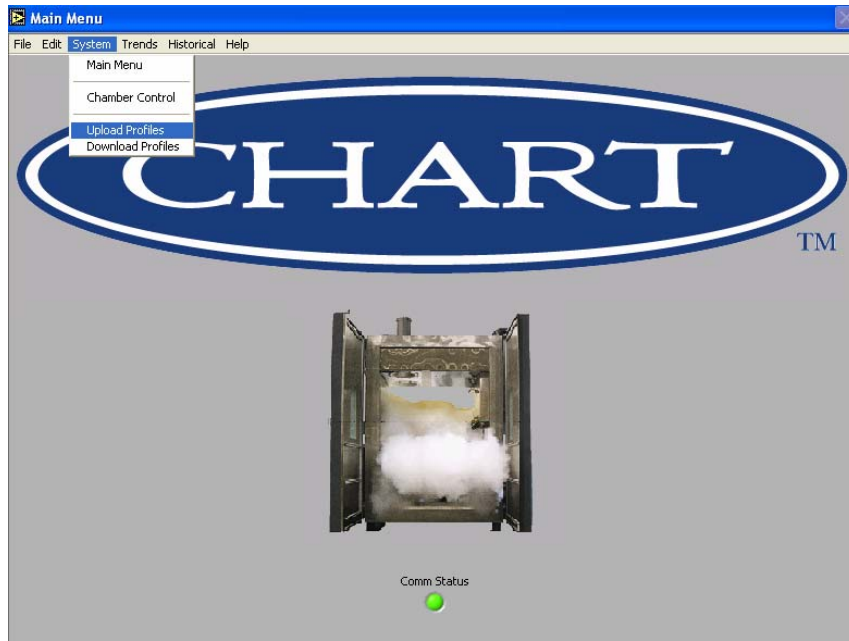


9. Watch the “Status” window for update messages. If there are any errors, they will appear here.
  - The downloading process should not take extremely long periods of time unless many profiles are being downloaded, or the profiles are exceptionally long. Typically, profiles require just a few seconds per profile for downloading.
10. A window will appear notifying the user when the download is complete.

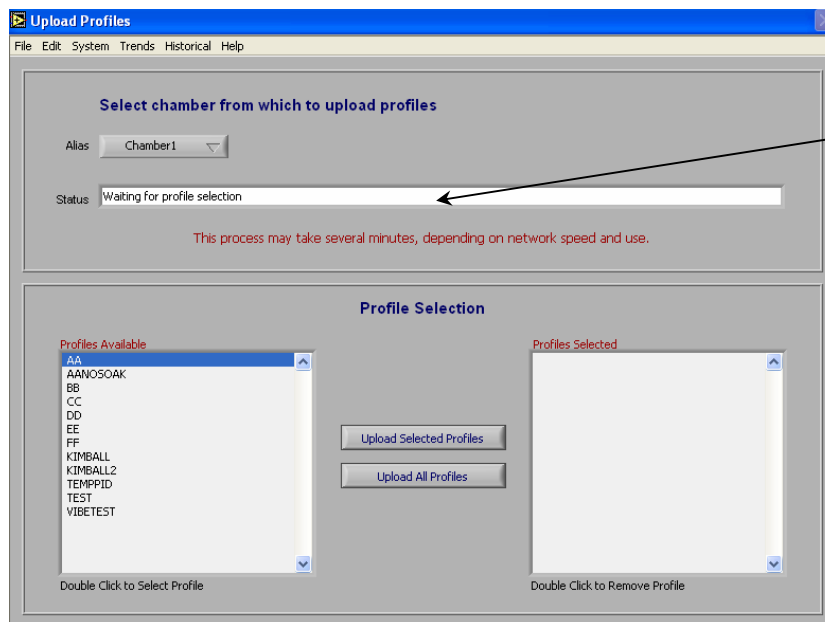
## Uploading Profiles (from PLC to PC)

The uploading utility can be used to transfer any profiles that are stored in the PLC’s memory to the PC. Once the profiles are on the PC, they can be edited, viewed, backed-up for future use, or downloaded to any other chamber. There must also be an active Ethernet connection between the PC and the PLC. The following procedure should be used to upload profiles from the PLC to the PC:

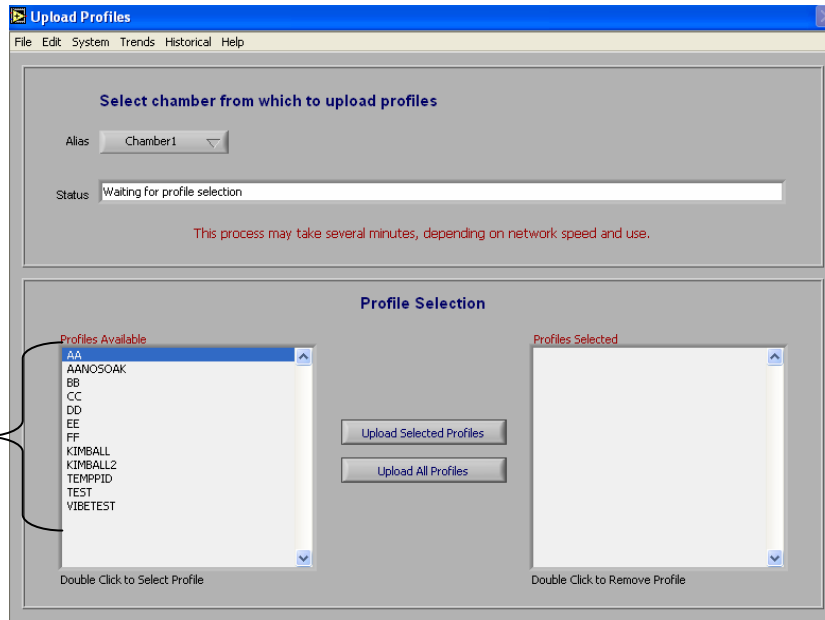
1. From the **System** menu, select **Upload Profiles**.



2. Verify there are not any error messages in the “Status” window.

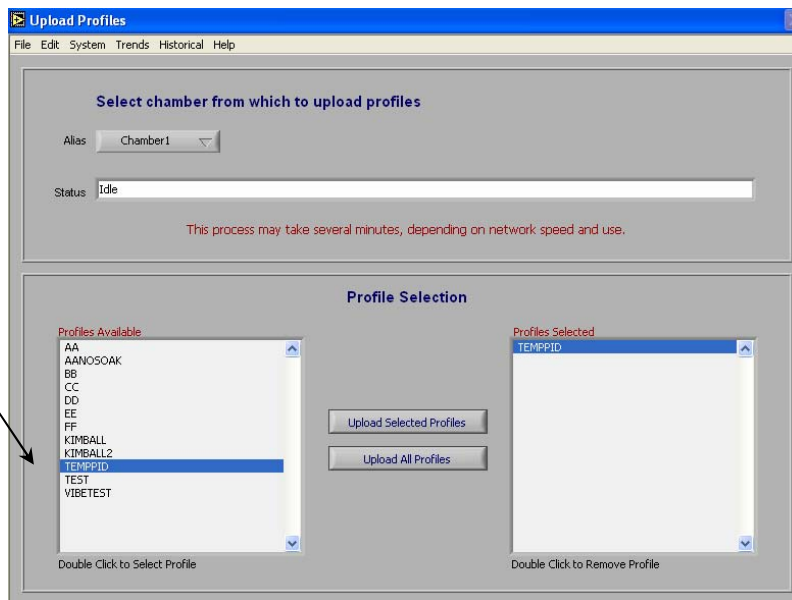


3. The screen will show a list of profile names that are currently saved in the PLC memory. If the profile name of interest is not shown in the “Available Profiles” window, then it is not saved in the PLC memory.



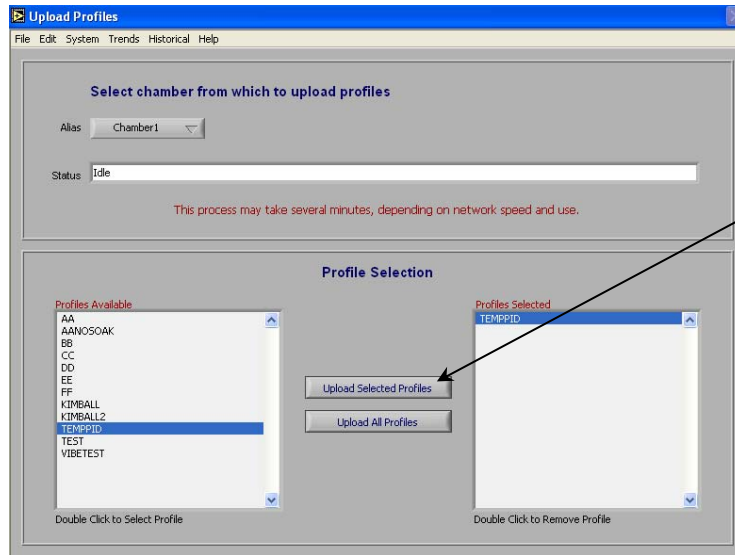
Profiles currently saved in the PLC memory will appear in this window.

4. Identify the profile that is to be uploaded to the PC.
5. Select the profile by double-clicking on its name.
  - Notice that when the profile is selected (by double-clicking its name), the name will appear in the “Selected Profiles” window on the right.



Similar to downloading profiles, as the profile name is double-clicked in this window, the name will appear in “Selected Profiles” window on the right.

6. Continue selecting profiles by double-clicking names until all profiles that are to be transferred appear in the “Selected Profiles” window.
7. If a profile is accidentally selected, it can be removed from the list by double-clicking the name in the “Selected Profiles” window.
8. When all profiles are selected, click the “Upload Selected Profiles” button.



9. Watch the “Status” window for update messages. If there are any errors, they will appear here.
  - The uploading process should not take extremely long periods of time unless many profiles are being uploaded, or the profiles are exceptionally long. Typically, profiles require just a few seconds per profile for uploading.
10. A window will appear notifying the user when the upload is complete.

## **Creating Custom Trend Charts**

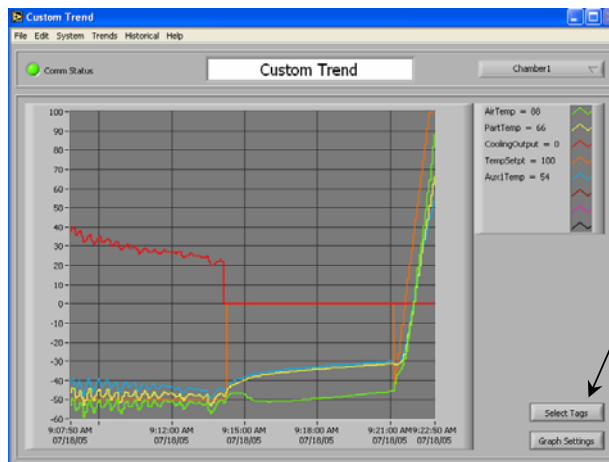
Custom trend charts can be created to develop a trend chart, which shows any combination of data the user would like to look at simultaneously. This allows the user to “filter out” data that is not of particular interest for this test, or to combine various data types (temperature and vibration) in any arrangement.

Use the following procedure to create a custom trend chart:

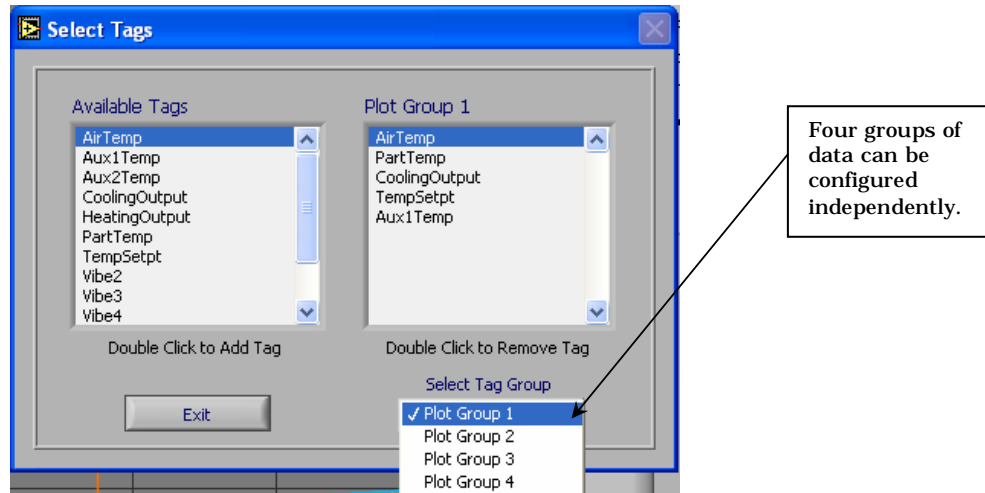
1. Select **Custom Trend** from the **Trends** Menu.



2. Select the data to be viewed on the chart by clicking the “Select Tags” button.



3. Select the group to modify the data points on.
  - The custom trend chart can have up to four groups of data defined at any one time. This allows the user to quickly look at one group of data for a particular test, then switch to a different group of data for another test.
  - For example, the groups could be configured such that:
    - GROUP 1 is temperature data
    - GROUP 2 is all vibration data
    - GROUP 3 is three temperatures and two vibration levels
    - GROUP 4 is three temperatures and three vibration levels



4. Select the data that is to be viewed from the list. Double click the name of the data you want to view in the “Available Tags” window to add that tag to the currently selected plot group.
  - Some of the data available are:
    - AirTemp – The temperature as measured by the thermocouple located in the chamber’s plenum.
    - PartTemp – The temperature as measured by the thermocouple located in the chamber’s workspace.
    - TempSetpt – The temperature set point of the chamber. If the chamber does not currently have a temperature set point, it will plot a value of zero.
    - VibrationLevel – The level of Grms measured by the control accelerometer.
    - VibrationSetpt – The vibration set point of the chamber. If the chamber does not currently have a vibration set point, it will plot a value of zero.
5. To remove unwanted tags from a plot group simply double click its name in the plot group window.
6. When a group is defined, configure the any of the other four groups as desired by changing the plot group from the “Select Tag Group” dropdown.
7. The groups are saved automatically as they are changed. When complete, select the plot group you wish to view and hit the Exit button to return to the custom trend window.

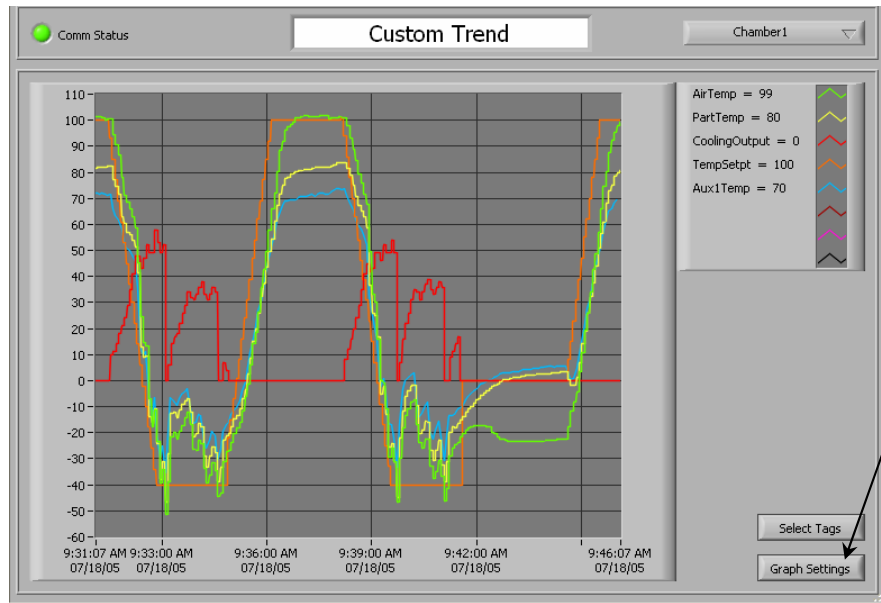
## **The Custom Trend Screen**

The custom trend screen was created to give users the flexibility to look at any collection of data together in groups, regardless of the data type (temperatures, vibration levels, set points, etc.). Once all of the data has been configured (see Creating Custom Trend Charts section for details), the look of the trend chart can be customized as needed.

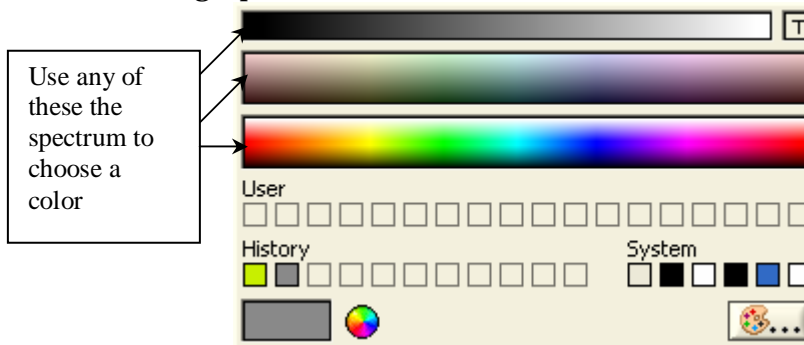
## **Changing the Trend Chart Background or Grid Color**

The main background color or grid color of the trend chart can be changed to any color. This can be done by the following procedure:

1. Click on the “Graph Settings” button.



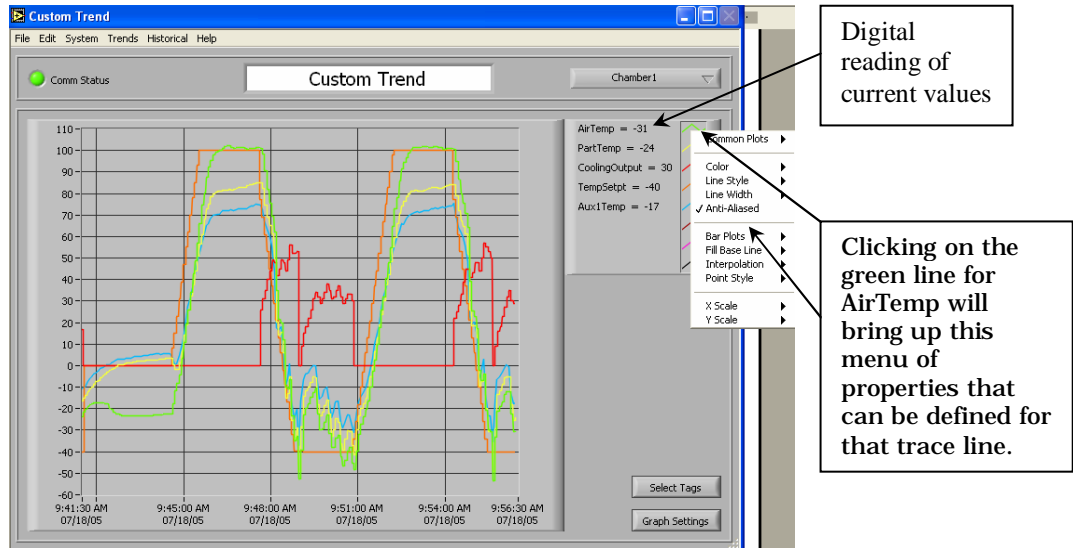
2. Clicking on the colored square representing the current background or grid color will bring up the color selection window.



To select a color move the cursor over the spectrum and click when the desired color is shown in the lower left corner. A recently used color can also be selected by clicking it from the “History” list of colors.

## **Changing the Trace Line Properties**

The properties of the individual trace lines can also be changed on the Custom Trend Chart. This is done by clicking on the line to be changed in the right hand legend.



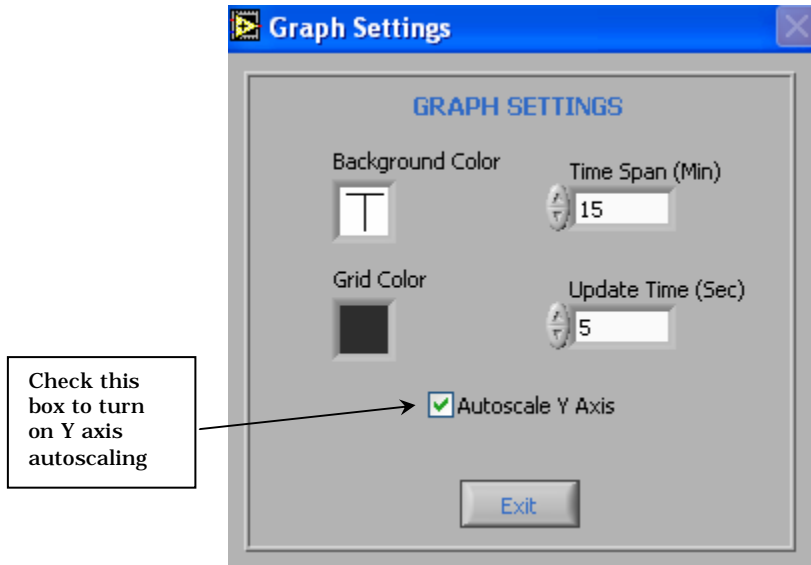
After clicking on the trace line of interest, a menu of properties that can be configured will appear. Select the property and define it as needed. Continue doing this for any and all trace lines to make the chart most appealing.

### **Changing Custom Trend Chart Title**

The Custom Trend Chart can be further customized to make the data appear just as needed. The title of the chart can be changed by clicking and dragging over the title to highlight it. Once it's highlighted, a new name can be typed in. When the new title is entered, do not hit the enter key! After the new title has been typed in, click somewhere on the chart to leave the field and have the new name take place. The title that was entered will become the new default title automatically.

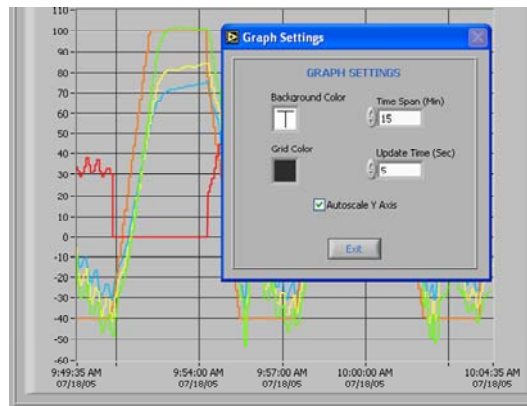
### **Changing Custom Trend Chart Scales**

The "Y-axis" scale of the Custom Trend Chart can be changed in two different ways. The first way is to use the auto-scale feature. With auto-scale turned on, the Y-axis will constantly update itself so that it fully shows all of the data. For example, if the temperature suddenly goes up to 100°C, the Y-axis will automatically make its maximum value something larger than 100, maybe 110 or 120 so that the new data can be clearly seen. The auto-scale feature will work in the same way on the minimum value. The auto-scale feature is turned on/off from the graph settings window:



If the auto-scale feature is not used, the minimum and maximum Y-axis values can be changed by clicking and dragging over the values on the graph to highlight them, then typing in a new value.

The X-axis (time) scale can be changed in the graph settings window. This value represents how many minutes will be shown on the full axis. In the following picture, the “Time Span” is set to fifteen minutes. The minimum time data (on the left of the axis) is 9:49, and the maximum time data (on the right of the axis) is 10:04, which represents fifteen minutes.

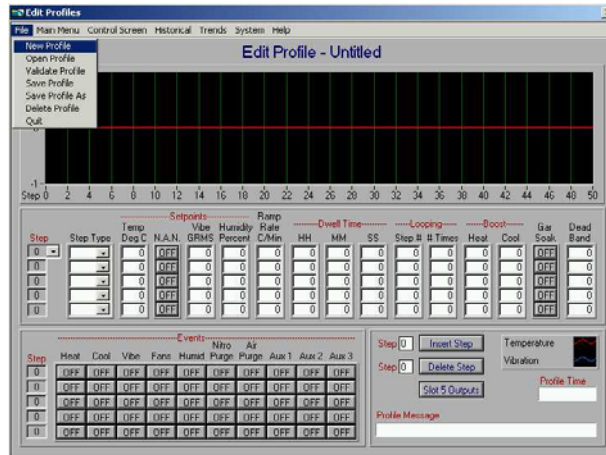


The graph update time is also set in the graph settings window. Chart recommends a value of 5 seconds to ease computer and communication resources.

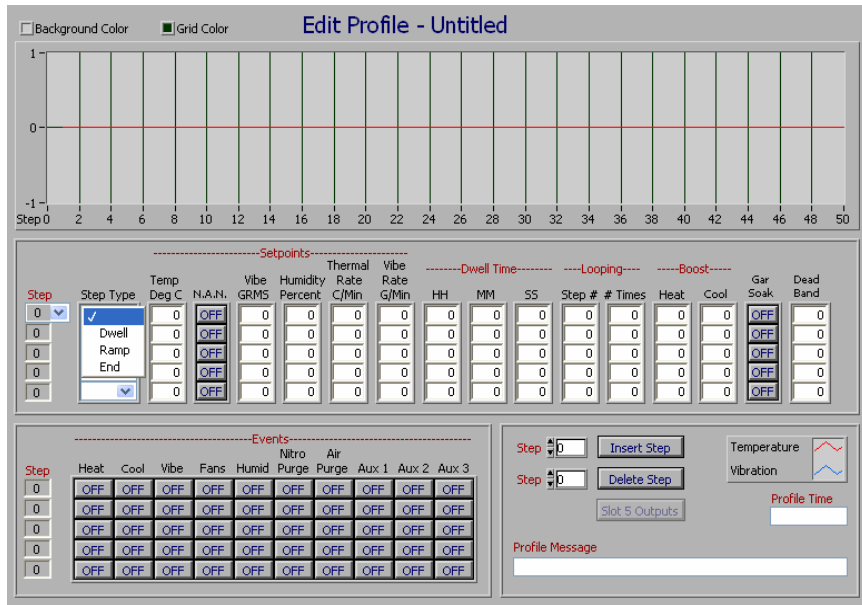
## Creating/Editing Profiles

The PC can be used to create new or edit any existing profiles much the same way as the touch-screen PLC. For details on the operation of various functions, refer to the manual for the CTI controller. To create a new profile:

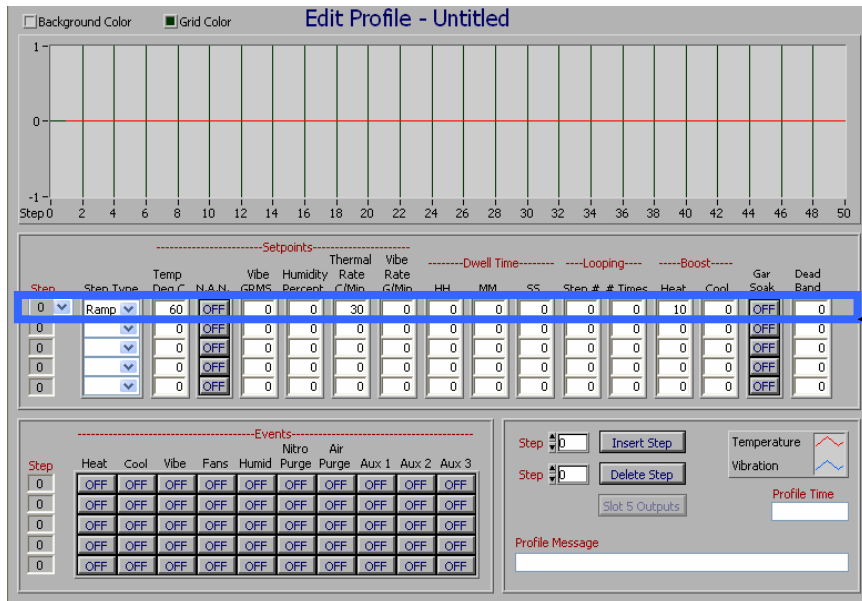
1. Select **New Profile** from the **File** menu.



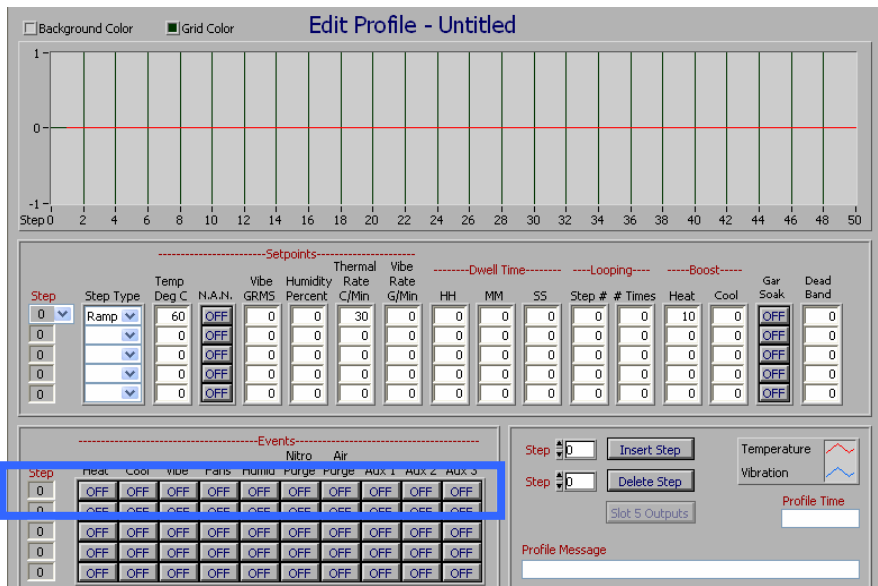
2. Start defining the profile by selecting the “Step Type” for the first step. The list of available step types is shown when the down arrow to the right of the box is clicked.
  - Note that when the first step type is defined, the step numbers one through five automatically fill in place.
  - Note that the next undefined step is automatically defined as an “End” step.



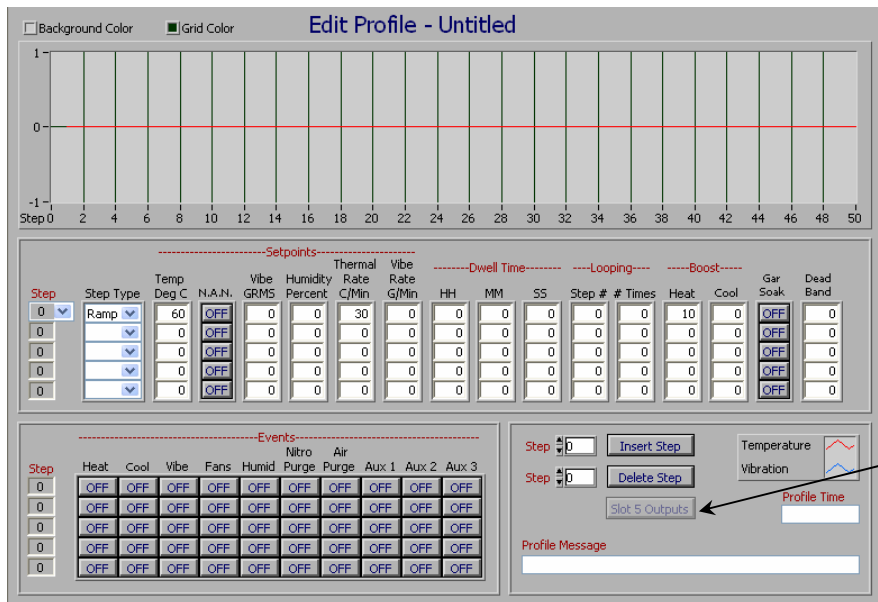
3. Continue defining all of the parameters across the row for that step.
  - Note that as set points are entered, lines will be drawn on the chart to visually represent the profile as it is created. This can help the user notice obvious data entry errors before running the chamber.



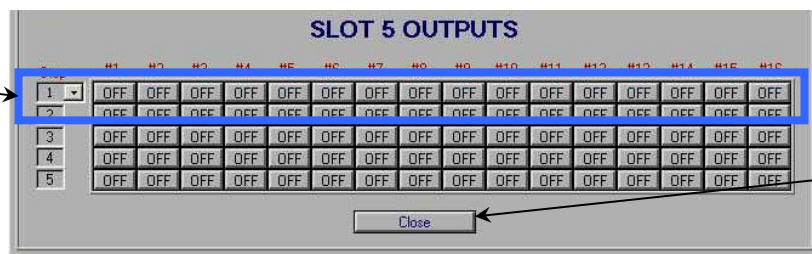
4. Define the state of the Events for that step by clicking the buttons between ON and OFF.
  - Be sure to note the correct step number is being modified. The step numbers that are shown in the top should match those shown in the Events section. For example, if step number three is being defined, be sure the Events being defined are also on the row for Event number three.



5. If the chamber was ordered with the additional discrete output relays, they can be defined for each step. Click the “Slot 5 Outputs” button to access the additional outputs.



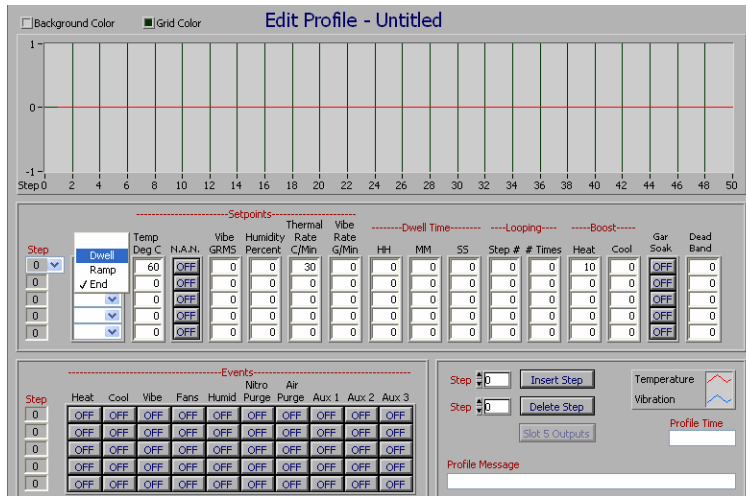
Define the state of the additional outputs. Pay attention to what step number is being defined!



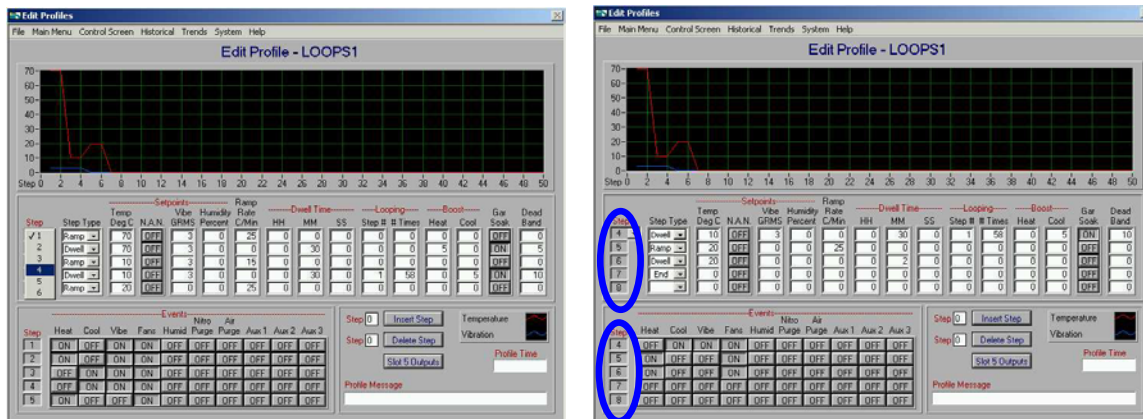
Click this button to define the additional discrete outputs.

When the outputs are defined, click the “Close” button to return to the profile definition page.

6. Continue by selecting the “Step Type” for the second step.



7. Continue defining steps and Events until the profile is completely defined.
8. If more than five steps are needed, additional steps can be shown by clicking the down-arrow next to the top step number. Then, select one of the numbers on the list. The number selected will now becomes the top step number and the following steps will fill in automatically.
  - Note that the step numbers for the Events will update automatically also.
  - Note that the step numbers for the “Slot 5 Outputs” will update automatically also.

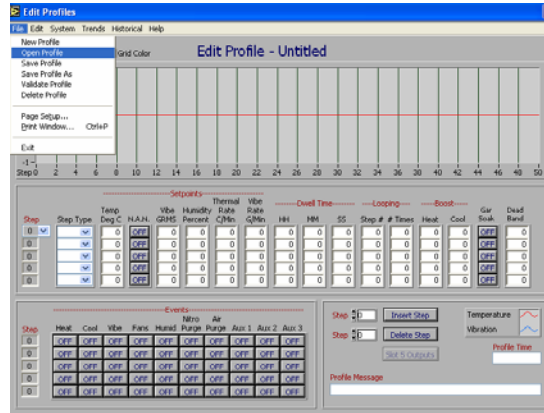


9. Save the profile by one of the following methods:
  - If it’s the first time the profile is saved, select **Save As** from the **File** menu to give the profile a name.
  - If the profile has already been saved, and the profile name is to remain the same, select **Save** from the **File** menu.
  - If the profile has been saved previously, but this version is to be saved under a different name, select **Save As** from the **File** menu.

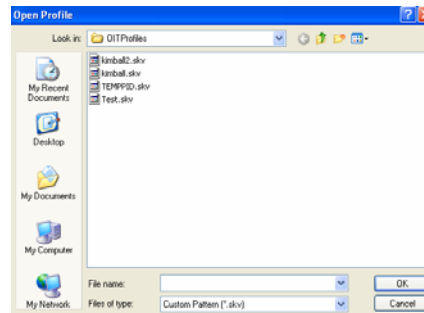
## Editing an Existing Profile

The software can also be used to make modifications to existing profiles. Once these modifications have been made, the profile can either be saved under the original name, or saved under a new name. To edit an existing profile:

1. Open the existing profile by selecting **Open Profile** from the **File** menu.



2. Select the profile name from the list by double-clicking the name.
  - Note that only profiles stored in the directory indicated in the window will appear in the list. If the profile is stored in another directory navigate to that directory to open.
  - Profiles that were created on the touch-screen of the CTI, and have not been uploaded to the PC, will not appear in the list.



3. When the profile has been pulled up on the screen, modify any portions of any steps as needed.
4. Save the profile.
  - Saving the profile with the same name will cause the computer to overwrite the original version of the profile.
  - If the original version of the profile should still be saved, give the new version of the profile a new name when saving by using "Save as".

## Profile Validation

While creating the profile, it is possible to do a profile validation check. The software has certain criteria that it knows must be defined in order for the

profile to run successfully on the chamber. Doing a validation check will tell the user if any of these fundamental rules have been broken. The software does a validation check automatically when the profile is saved, but it is also possible to do one at any time of the profile creation process by selecting **Validate Profile** from the **File** menu.

If the software detects an error in the profile, it will give the user a message on the screen describing the error, and inform which step has the error. When this happens, go to that step and correct the error. When the error has been corrected, try validating or saving the profile again. If there is another error, the software will again flash a new message. If the profile passes the validation check invoked by trying to save the profile, the profile will be saved.

The screenshot shows the 'Edit Profiles' window with a graph at the top and a configuration table below. The table has columns for Step, Step Type, Temp, N.A.N., Vibe, Humidity, Thermal Rate, Vibe Rate, Dwell Time, Looping, Boost, Heat, Cool, Gar Soak, and Dead Band. Step 1 is set to 'Dwell' with a Temp of 0 and Dwell Time of 0. An error message 'ERROR IN STEP 1' is displayed in a red box, with an arrow pointing to the 'Dwell Time' field for Step 1. Two callout boxes are present: one pointing to the 'Dwell Time' field for Step 1 with the text 'Which step number had the error.', and another pointing to the error message box with the text 'Brief description of the error.'

In the above example, an error was found in step #1 because no dwell time was entered for a dwell step.

## Collecting Data

The software can be used to create data files for future examination. These data files will be delimited data files that can be opened in any common spreadsheet application. The following procedure should be used to set up a data file:

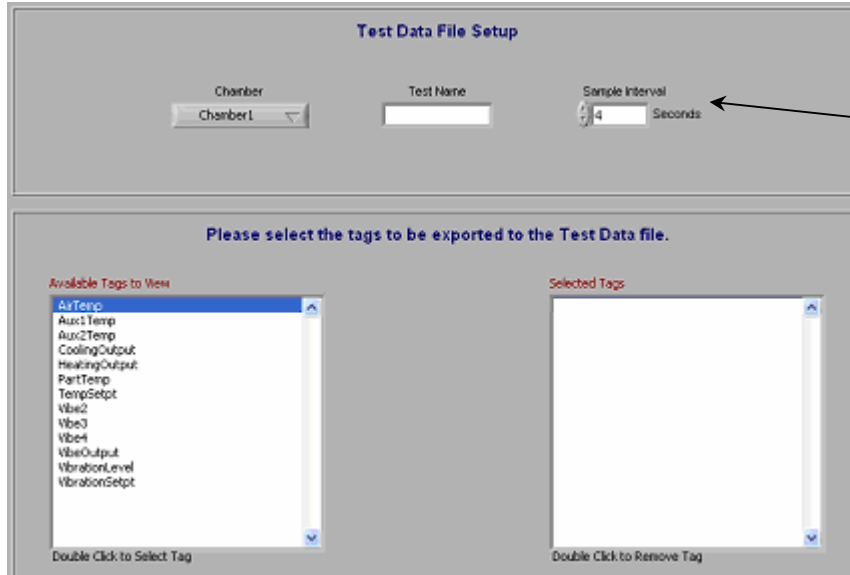
1. Select **Historical Setup** from the **Historical** Menu.



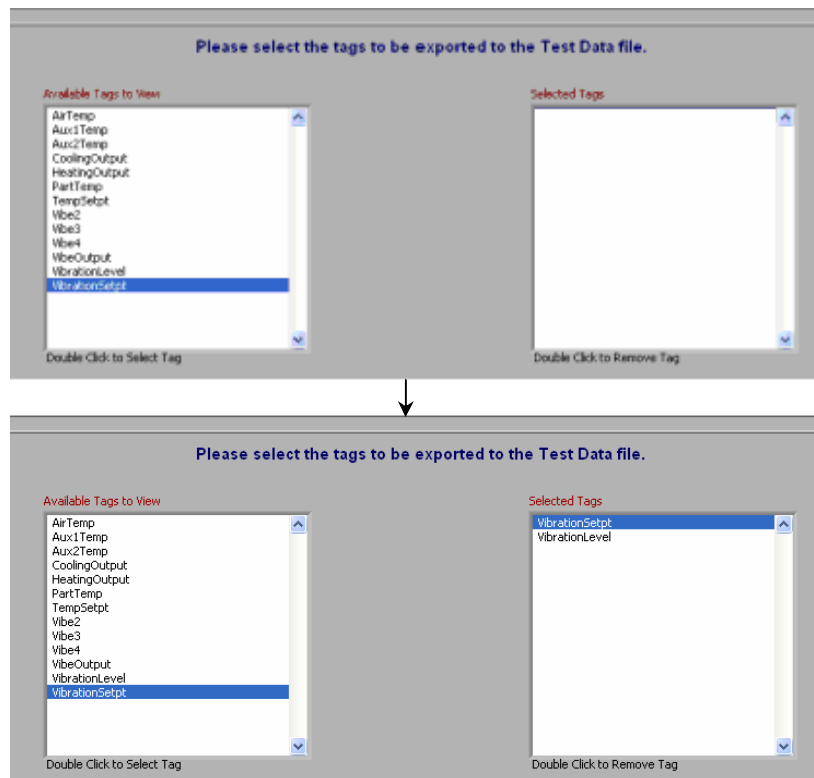
2. Assign a data file test name by clicking in the box and typing a name.
  - The data file will be saved as a file with a name, which begins with what is typed in this box, and ends with a time/date stamp.



3. Define the time interval (in seconds) between data samples.



4. Select the data (“tags”) to be logged for this test by double clicking on the name in the “Select Tags to View” window on the left.
  - When the tag has been selected, its name will appear in the “Selected Tags” window as seen below.



5. If a tag is accidentally selected, it can be removed from the list by double-clicking its name in the “Selected Tags” window.

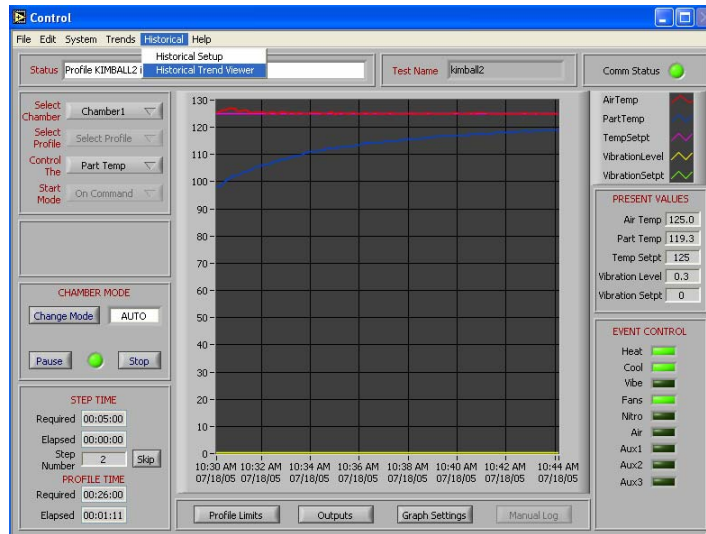


6. Unless this setup is changed, all future tests will log the same data tags at the same rate, with the same text name for a file name. The data files will be distinguishable by the time/date stamp in the file name.
7. The data files will be stored in the C:\Program Files\National Instruments\LabVIEW DSC Run-Time System 7.1\data directory on the hard drive.

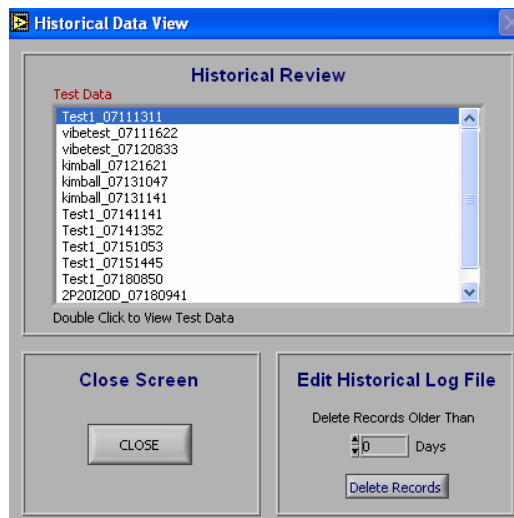
## **Reviewing Data**

Data already stored in the database can be reviewed or exported at any time through the historical data viewer.

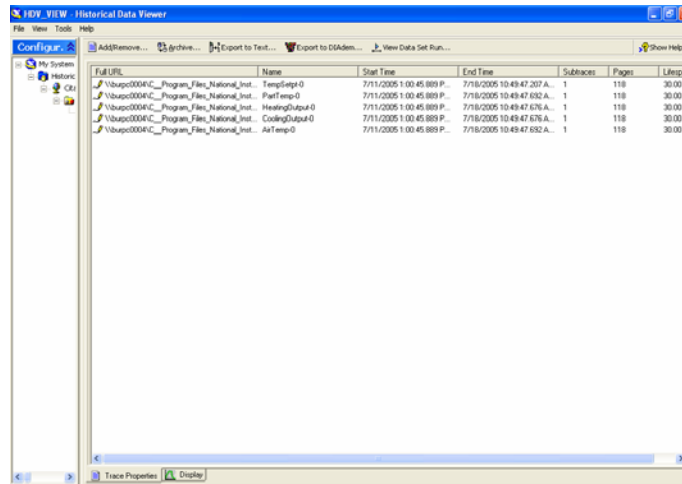
1. Select "Historical Trend Viewer" from the historical menu.



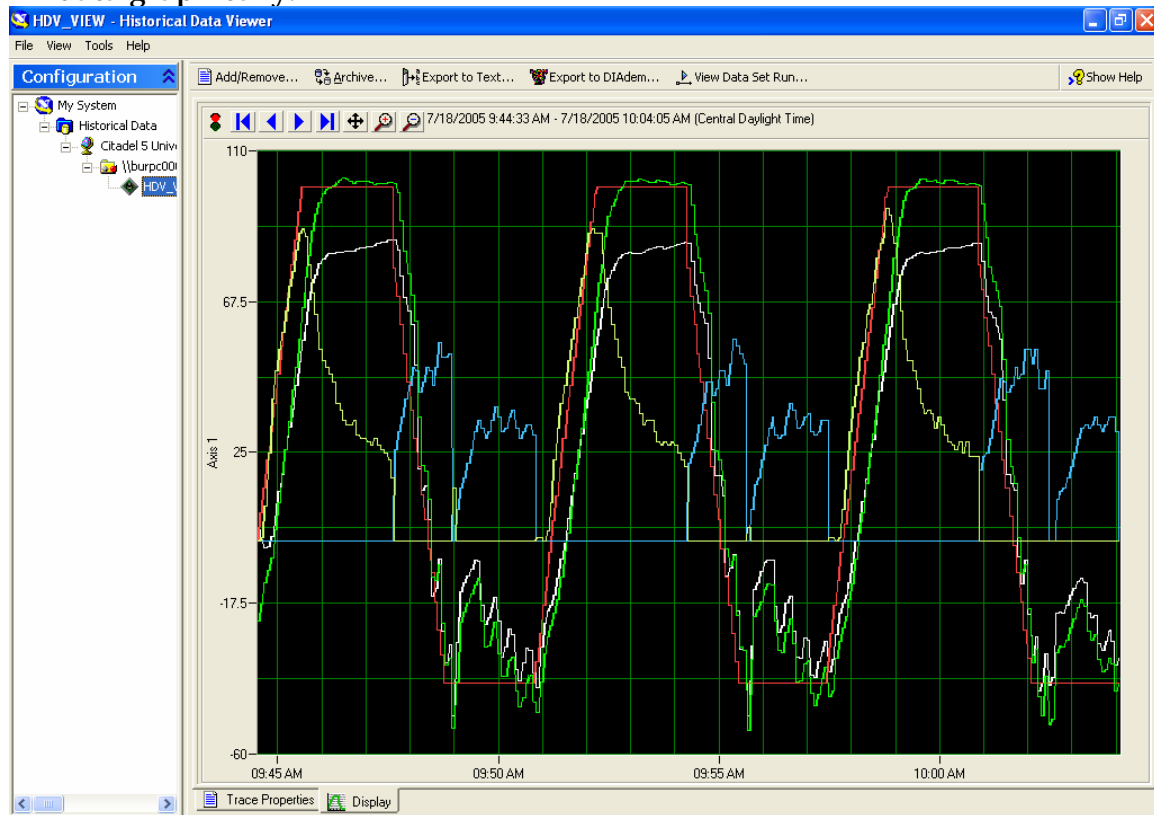
2. A window containing a list of all the data files will open. From this window select the set of data you wish to view.



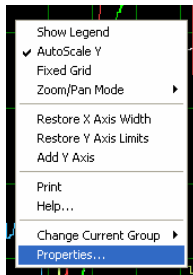
3. A window will open that shows the properties of the saved data. Selected tags, start time, and stop time is shown in this window.



- Click the “Display” tab in the lower left corner of that window to view the data graphically.

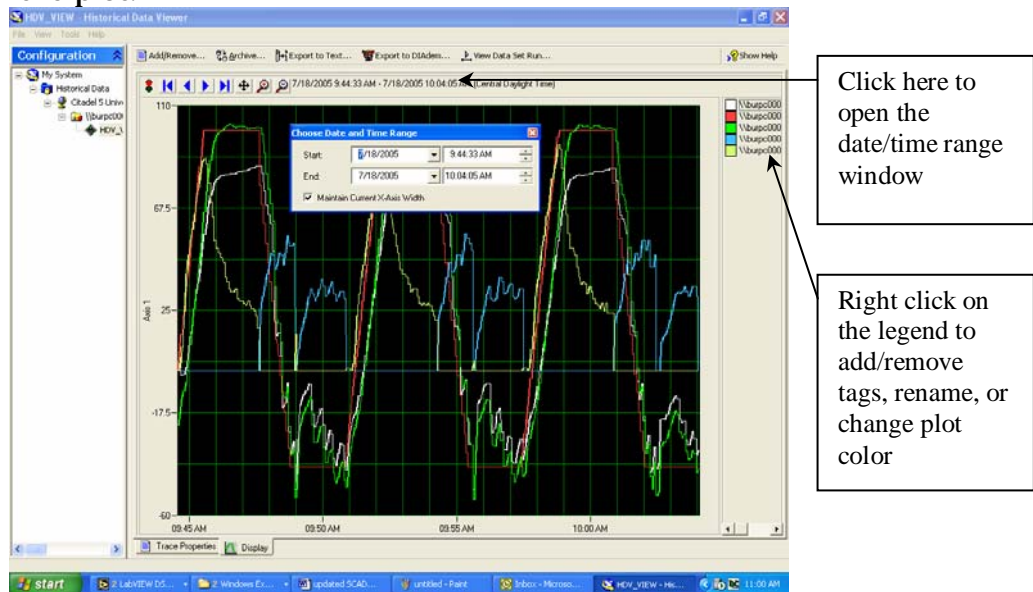


- Right-clicking on the graph area will bring up a menu containing many options about the graph.



- “Show legend” will open a legend on the right to show the color of each data channel. Clicking the colored box next to the name will allow you to change the color of that variable on the plot.
- “Autoscale Y” allows you to turn on/off the autoscaling feature for the Y-axis.

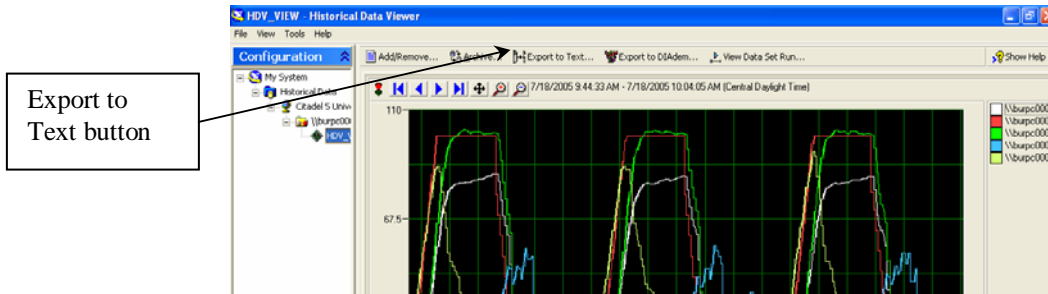
6. Clicking on the time will bring up a window to set the total amount of time shown on the plot.



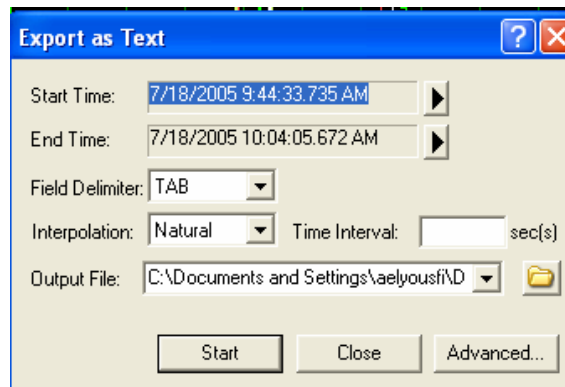
## Exporting Data

The historical trend viewer allows you to export anywhere from a short segment of the data to the entire file for use in any spreadsheet program.

1. To export data click the “Export to Text” button. The “View Data Set Run” button is used to view data that has been logged for a pre-defined group of tags. Since tags are already selected from the Historical Setup window this option is unneeded. The “Export to DIAdem” button is used to export data to the National Instruments DIAdem data analysis package. This package is not needed with the SCADA software and is therefore not used.



2. A window is opened to specify the exact data to be exported.



- “Start time” is the desired beginning time of data to be exported. Clicking the arrow next to the current selection can change this value.
- “End time” is the desired ending time of data to be exported. Clicking the arrow next to the current selection can change this value.
- The “Field Delimiter” dropdown allows you to select how the output file will be delimited.
- The “Output File” dropdown allows you to set where the file will be created. Click the folder icon to the right to browse your entire system.

3. Once the desired data and options are selected simply press the “Start” button to export the data to the selected file. From there you may open that piece of data with a spreadsheet program of your choosing.

**Export data from manual mode operation:**

If the chamber is running in manual mode and you want to export a piece of data, simply open the historical viewer from the “Historical” menu and open the most recent set of data. Pressing the red/green stoplight button in the upper left will switch to live mode. This will show the current chamber data and at any time you can export this data using the “export to text” button.