Performance Testing

User's Guide

Table of Contents

Software Features...

The Performance Testing system is designed to help you test the performance of a natural gas compressor.

You can acquire and process measurement signals in real time, or analyze your existing measurements.

Flow measurements can be made using a variety of flow measurement devices, using the American Gas Association (AGA) calculation methods.

You can create Wheel Maps based on adiabatic, isentropic, or polytropic head, and efficiency, and display lines of constant gas power, as well as surge limit and surge control lines. Gas compression properties are calculated using the AGA-10 equation of state.

You can also calculate the capacity of recycle or anti-surge control valves (and their associated piping).

After a test, you can recalculate the results based on new gas analysis results, or new transducer calibrations, and save the results as a spreadsheet file for further analysis.

A variety of views of your real-time measurements are available:

- "schematic diagram" of the individual units.
- strip charts
- Wheel Maps
- X-Y graphs
- etc.

The software also includes an extensive on-line help system.

Getting Started

Connecting to the system

The Performance Testing software is pre-installed on your data acquisition unit. You can connect to it in two ways:

1. By hooking up a monitor, keyboard, and mouse to the appropriate connectors
2. By connecting an ethernet (T-base 100) cable to the network connector on the front panel connector (RG45).

In most cases, it is more convenient to connect as a remote Windows session. You may wish to connect directly, if you are unable to connect remotely.

Using a Remote Windows Session (Windows XP) or Windows Terminal Services client (Windows 2000), connect to the system. The computer name is *GasPerf*.

For your initial login, use:

User name: administrator
Password: Andromeda

You can create additional User names to suit your own needs. By default, the GasPerf system has been assigned a static IP address within Andromeda's assigned IP range, usually 198.161.125.100. You can change this to an IP address in the same subnet as your laptop computer.

To start the system, double-click the "Performance" icon on the desktop.

Starting the software

To start the Performance Testing software, double-click the icon for GasPerf.exe (or any shortcut to the file). After a few seconds, the main menu panel appears:
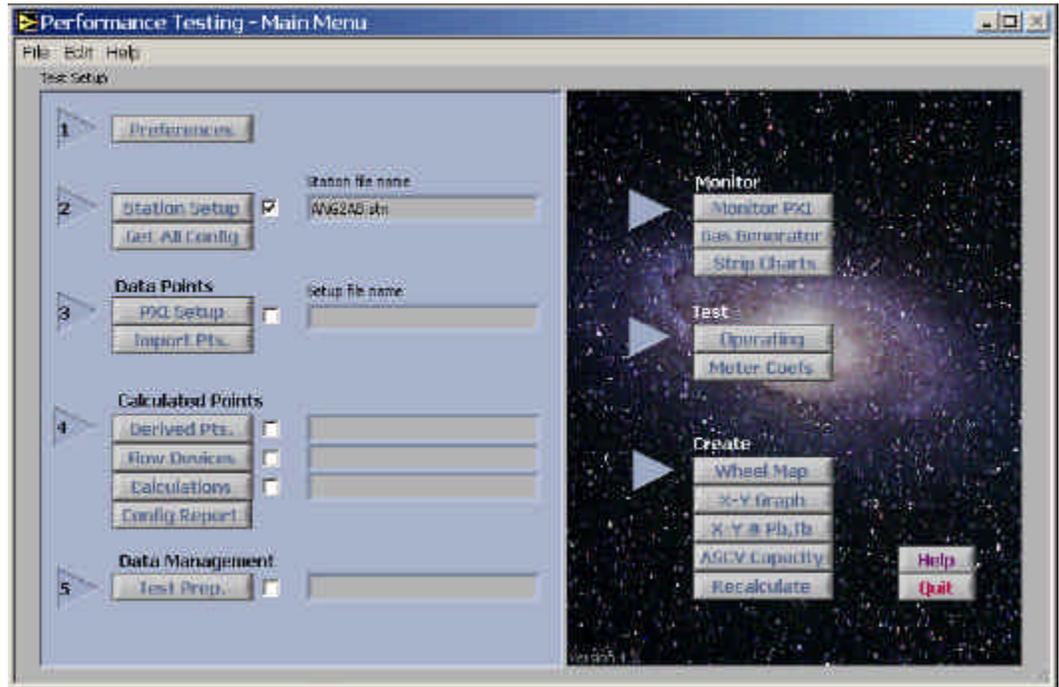
While the cursor is in the main part of a window, it is in the shape of a pointing hand. You can select a function by clicking the appropriate button.

When you select a function, its panel appears. This panel will have a "Done" button on it somewhere. When you're done with the panel, click its Done button to return to this menu.

You may not need this Guide...

It's not quite true that you can throw this manual away, but the software does have an extensive on-line help system. Here's how to use it:

Full-Screen Help

Most panels contain a "Help..." button. Clicking it displays general information about using that panel (you can usually press the F1 function key instead of clicking the Help button). The help panel may contain buttons to display related information.

There may also be a pop-up menu in the upper left hand corner of the screen, which lets you select from a variety of help topics.

Every panel also has a "Done" button — click it to return to the previous panel.

Floating help window You can more the Help window around the screen, and resize it. When the cursor is over a screen object, the help window displays information about that object. Select "Hide Help Window" from the Help menu to remove the help window.

Tip strips If you let the cursor hover over a field, a tip strip give you a short summary of what the field is for.

Setting Up

Let's jump right in by configuring the PXI chassis for a test. We'll go through the setup steps in the main menu in order:
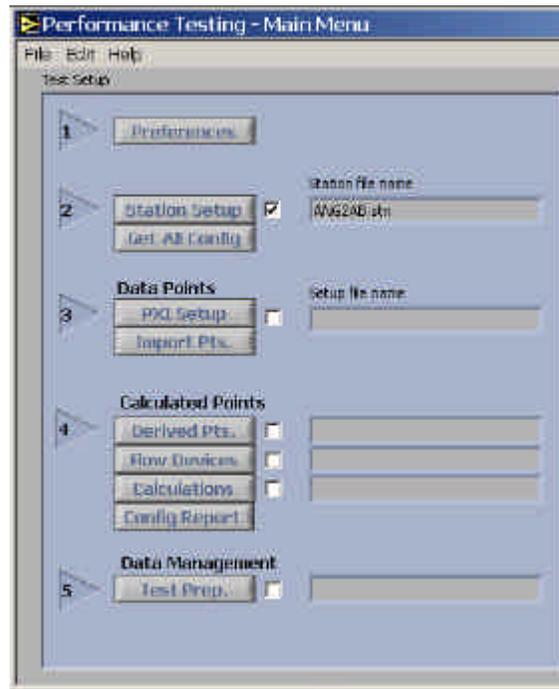
Note that the steps are numbered, and that several of the buttons have a checkbox and a file name just to the right of their button.

Although you don't have to go through the setup steps in order, each steps builds on the previous step.

When you go through a setup step, a checkmark appears beside it, and the appropriate file name is shown.

Preferences

File Preferences

If this is the first time you've used the system, you should tell GasPerf where you'd like to store your data files. Do this by clicking the [ Preferences] button on the main menu, or selecting "GasPerf Preferences" from the menu bar[1] . The Preferences vi appears (the values shown are the default values you get if you click "Default"):
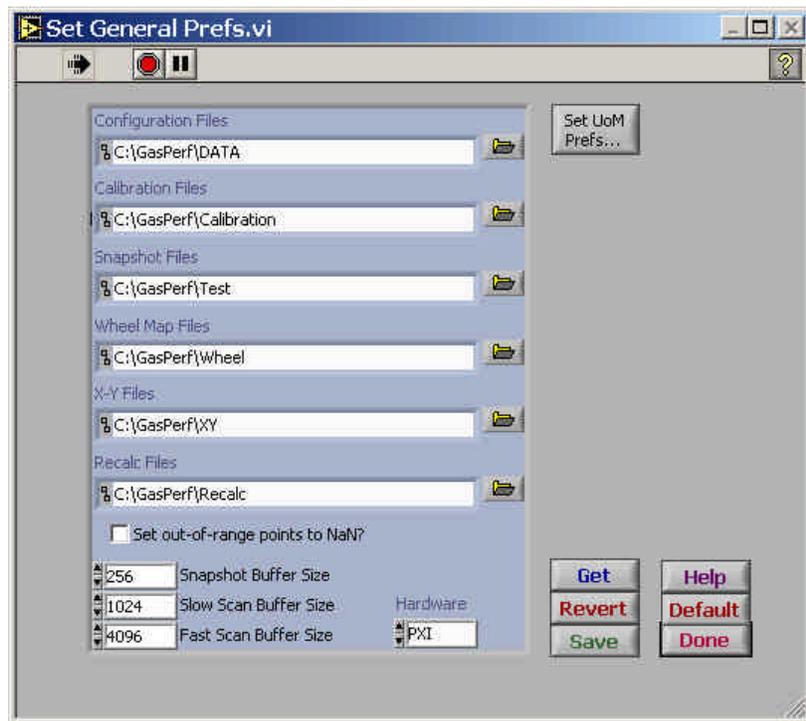
The Performance Testing system uses several types of data files. For each of these files, enter the *path* to the folder on your hard disk where you'd like them saved. When you retrieve or save a file, the file dialog will start at this location, but you can always specify a file located somewhere else.

- There are a couple of shortcuts you can use instead of typing in the full paths:
- Drag and drop a folder from the Windows Explorer.
- Clicking the "folder icon" at the right of each line
- Copy and paste by:
  - selecting some text in one of the boxes
  - right-click the mouse, and select "Copy"

---

[1] Only available in some screens.

- position the insertion point in the destination box, right-click the mouse, and select "Paste".

Click "Save" to save your file preferences in a file called *Prefs.dat*, located in the same folder as the Performance Testing application. You cannot change the name or location of the file (if you could, how would the software find it?).

"Revert" restores the paths that were in *Prefs.dat* the last time you saved them.

"Done" allows you to use these path names for the remainder of this session, but does not save them permanently.

Please note that when you define a Station Setup, the location of the configuration files is changed to the directory where the station information was stored or found.

To change the unit preferences, click this button:

Unit Preferences

The Performance Testing software always stores both measured and calculated information in SI units, but you can usually information in your choice of SI units or U.S. Customary units of measure.

The *Unit Preference* vi lets you specify the units of measure you prefer to see:
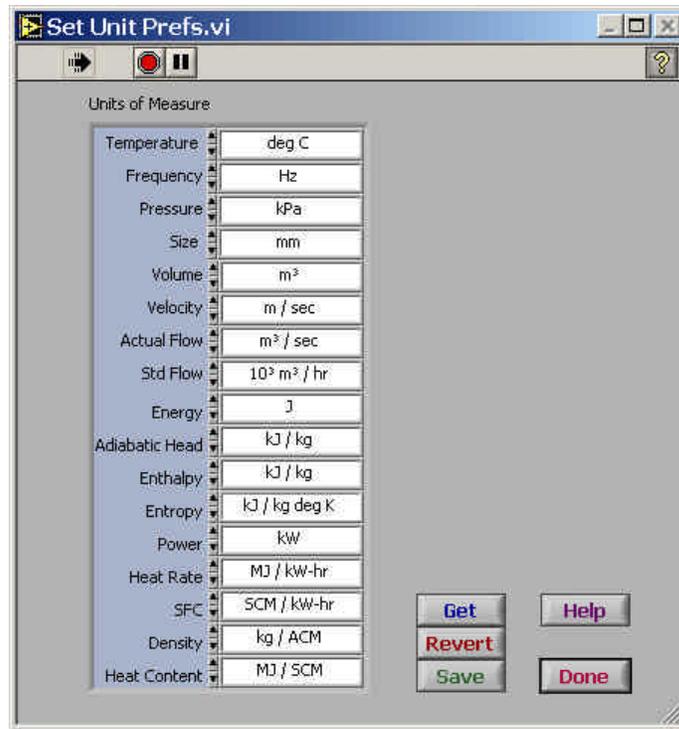
You can specify each of type of units from its pop-up menu, using any combination of SI and U.S. units.

The buttons work like their counterparts in the File Preferences:

"Save" stores your unit preferences in a file called *Units.dat,* located in the same folder as the Performance Testing application. You cannot change the name or location of the file.

"Revert" restores the paths that were in *Units.dat* the last time you saved them.

"Done" allows you to use these unit preferences for the remainder of this session, but does not save them permanently.

Station Setup

The information which relates to an entire station, or to an individual compressor unit is defined by "Station Setup":
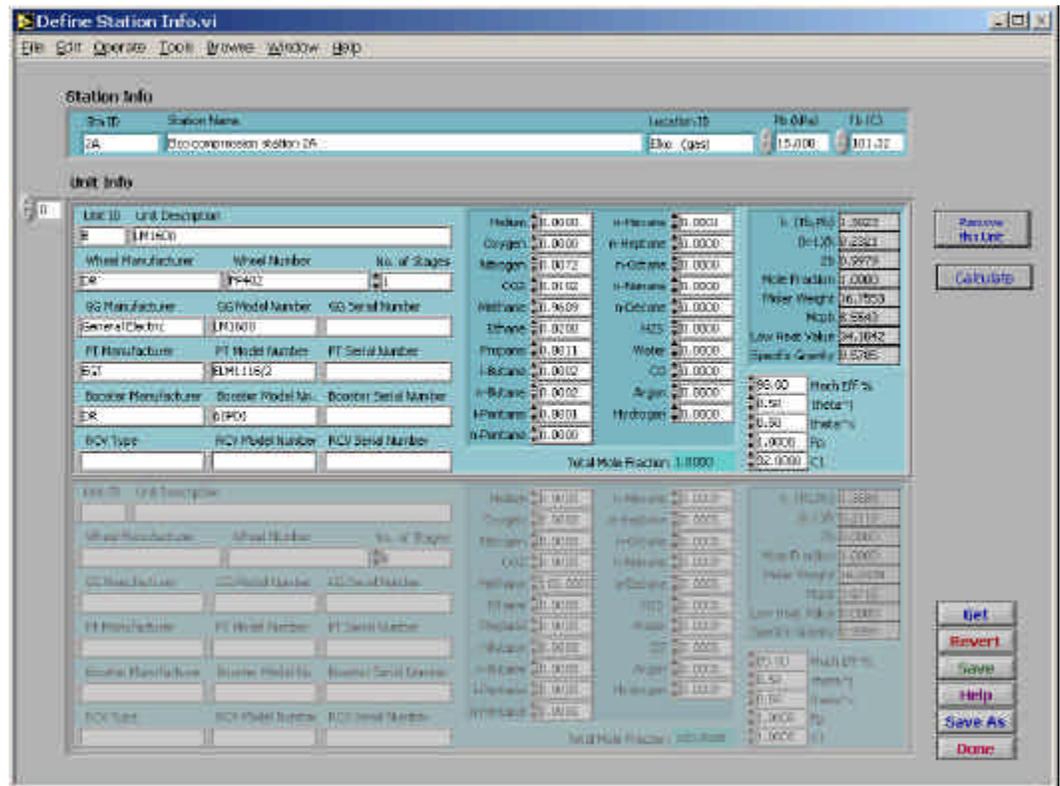

The base pressure and temperature are used to define such items as "standard flow", and must be entered in kPa and degrees C.

The lower portion of the screen lets you define individual compressor units. You give each unit a unique *Unit ID*, which will be used as part of the tag name associated with each signal. Although the Unit ID can be any length, we suggest you limit them to one or two characters if possible, so that the tag names don't get excessively long. You can define any number of units.

The gas composition is entered in mole fractions or mole percentages. Be sure to click the *Calculate* button after you change the base conditions or gas composition, to update the calculated gas constants on the right-hand side of the screen.

Some other information you need to supply is:

Mech eff %	The mechanical efficiency of this compressor unit. Typical values are 95% for liquid-film (oil) bearings, and 97.5% for magnetic bearings.
theta^j	The temperature correction factor (theta) is raised to this power for speed corrections (ie: typically the correction divides the uncorrected values by theta^0.5).

theta^x   The temperature correction factor θ (theta) is raised to this power for shaft power corrections:

$$PWRs.isen.c = \frac{PWRs.isen}{\left(d \cdot q^{x}\right)}$$

where

$$d = \frac{Pa}{Pb}, \text{ and } q = \frac{Ta\langle Kelvins \rangle}{Tb\langle Kelvins \rangle}$$

Fp   The piping geometry factor:

$$\frac{LineSize_{inlet}}{ValveSize_{inlet}}$$

and then interpreted from a table

C1   ASCV coefficient "C1", as supplied by the manufacturer (defined as the ratio Cg/Cv).

Important Note   If you remove a unit .by clicking "Remove This Unit", signals which refer to the missing unit will be assigned to the next unit — so be sure to check the rest of your configuration to make sure the units are still correct!

Remember that if you forget what a field is used for you can display the floating help window (Ctrl+h).

Get All Configuration Files

If you've already defined a complete set of configuration files, you can retrieve them together at this point by clicking *Get All Config* in the main menu:
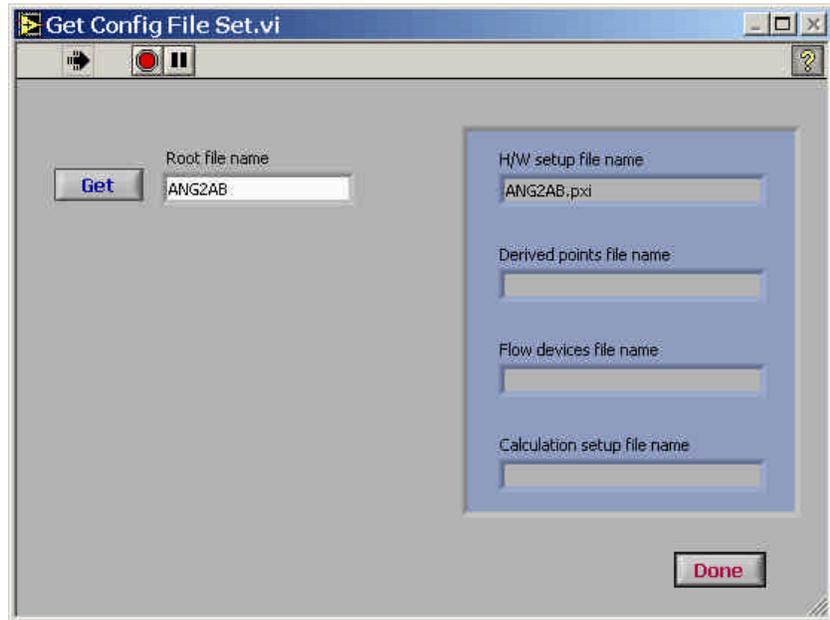
This vi will initially attempt to load all of the configuration files which start with the station name (as shown above).

If you want to retrieve all the files which start with a different name, change the root file name and click "Get". If all of the files have different names you're best off to visit the individual configuration screens shown in the next few sections of this Guide.

About Tags

Tag Names

Every measurement or calculated value in the Performance Testing system is given a *Tag Name*.

Tag name are short identifiers (typically three to six characters) used to identify a particular measurement.

Each tag name is associated with either a particular unit, or the overall station. For example, if we are working on unit M5, the tag "M5:Ps" might refer to the suction pressure measurement for unit M5.

Once again, every tag name used in the Performance Testing system is qualified by either a unit name or "Stn" (for station-level measurements).

If it's possible to just use unqualified tag names, there will be a pop-up menu nearby, from which the unit name can be selected.

You are free to use anything you want as a tag name, provided you do not use the colon or space characters. If you type in a qualified tag name, separate the unit name and the tag name with a colon, but no spaces.

If you use particularly long tag names, they may not fit into the tag name fields on the screen (in some cases they will appear blank). This does not affect the operation of the system, but can be an inconvenience.

Signal / Tag Order

You can often select tags from pop-up menus. These always appear in the same order in which measurements are acquired and calculated. The recalc file is also in the same order.

The order is as follows:

- Frequency / time signals, ordered by connector number.

- Thermocouples, in connector order.

- RTDs, in connector order.

- Analog signals in connector order (self-powered loops at the end).

- Manual inputs, in the order you defined them.

- Shared signals, in the order you defined them.

- Averaged signals, in the order they were defined.

- Flow devices, in the order they were defined, within the order of the meter type tab in the "Define Flow Devices" vi. Two additional tags, *tag.fpv* and *tag.z* are automatically added after each flow tag.

- Sum/Difference signals.

- Equation of State calculations, in the order in which they appear in the "Define Calculations" vi.

- ASCV capacity calculations.

Hardware Setup

You can acquire four types of signals from the data acquisition system:

- Frequency or time measurements
- Thermocouple temperatures
- 4-wire platinum RTD temperatures
- Other current loop analog measurements (passive or self-powered)

You select which of these you wish to configure by selecting the appropriate tab.

All four types of PXI setup information are saved in the same file. By default, the system will propose *Station name.PXI* as the name for this file, but you can use any file name you wish. By default, only files with the extension .PXI are visible in the open and save dialog boxes.

Frequency and time

Frequency measurements are made using a National Instrument 6602 counter/timer card. You can test for the presence of signals using the Measurement and Automation Explorer, if you wish.

Unlike some other frequency measurement hardware, it's important to understand that the PXI 6602 card can *only* measure TTL signals. This means that you may require external signal conditioning, such as a Schmidt trigger, in addition to any isolation.

Frequency measurements are more complex than the other analog signal measurements, so let's look at the configuration in more detail:

This example is a configuration for the "F3" connector on the front panel. The Module and Channel correspond with the PXI module name and the PXI-6602 counter channel in MAX (the National Instruments Measurement and Automation Explorer).

| Connector Name	The *cluster* displayed for each frequency connector shows the connector name, and the module and channel numbers. You can't change them (if you try, they will be restored to their correct values the next time you display this window).
Scan?	Check this box if you want this signal to be acquired. The tag name also needs to be non-blank for a signal to be part of the scan.
Corrected?	Check this box if you are measuring a speed, and you wish to correct the speed for ambient conditions. The corrected speed will appear as a new tag with the same name as this tag followed by a suffix of ".c".
Unit ID	This pop-up menu lets you associate the measurement with the entire station (Stn), or with a specific compressor unit.
Tag	Each measured or calculated value in the system has a "Tag" — a short identifier used to refer to the signal.
	Tags can be qualified, or can refer to the current unit. A qualified tag has the form:
	UnitID:Tag
	For more information about tags, please refer to the "About Tags" section of this Guide.
Description	You can enter a text description about the measurement you are making in this field. The description can be any length, although only the part of it can be shown in this box.
Sensor ID	Each sensor can have an identification tag, which you can make a note of here. This is just a comment field — the system does not use this field to, for example, diagnose faulty sensors.
Location	You can make notes about the measurement location here.
Trigger	You need to specify either rising edge or falling edge.

Min exp.	You need to enter the minimum expected frequency or time period here (the acquisition won't work if you don't). Frequencies or time periods outside this range can't be measured.
Max exp.	You also need to enter the maximum expected frequency or time period here (the acquisition won't work if you don't). Frequencies or time periods outside this range can't be measured.
# Samples	You need to tell the hardware how many signal edges to measure and average.
Meas time	You need to tell the system how much time to allow for the measurement to complete. If you set it too low, you will get measurement errors. If you set it too high, it may take a while for the system to report that a signal can't be measured. In case of doubt, we'd suggest erring on the high side.
Raw Units	The frequency or time period that you measure can be scaled. The measurement is a "raw" value in Hz or seconds. Enter any convenient range of raw units
Scaled Units	The "scaled" value can be in units:
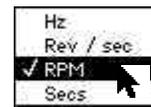
If you don't wish to scale the measurement you made, set the Raw and Scaled range to be the same.

If the measurement falls outside of the range you set here, it is set to "NaN" (Not a Number), and a range error is signaled.

Thermocouples
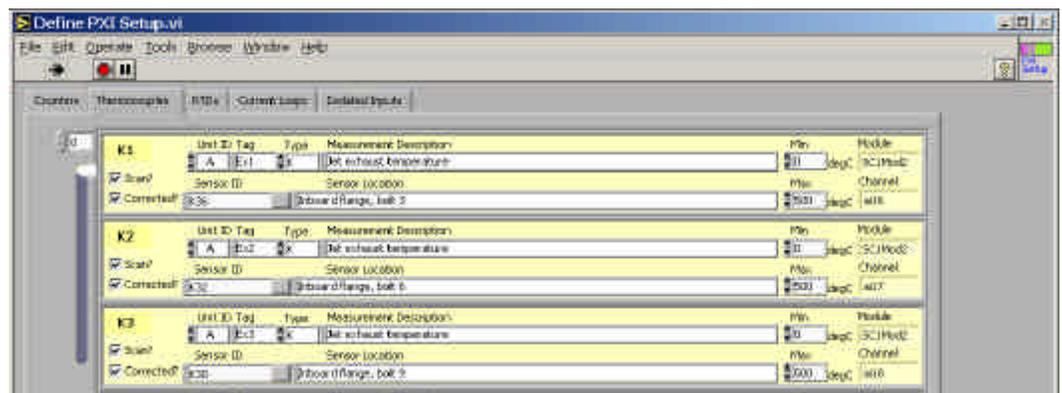
Since quite a few thermocouples can be used, you can use the vertical slider to move quickly between them.

Let's look at the configuration in more detail:
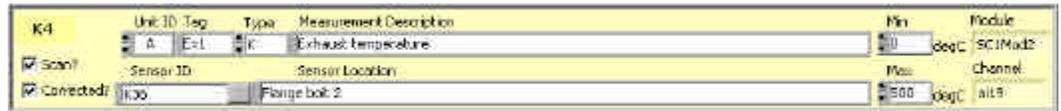
The thermocouple configuration is a bit simpler than the frequency / time configuration. There is no section for scaling to SI values, because the temperatures are always converted to degrees C.

The new fields are:

Type
: You can select from any of the thermocouple types which are supported by the PXI hardware. In practice though, the internal acquisition chassis wiring is of type K, so that's the only kind of thermocouple you can read without additional cold junction temperature compensation.

Min
: The minimum allowable temperature, in the temperature units you specify to the right o the field. Note that you can right click on the units of measure field for help in specifying other units of measure.

Max
: The maximum allowable temperature, in whatever temperature units you specify to the right o the field.

Module/Channel
: The SCXI chassis number (always 1) and module number (2 for the thermocouples and RTDs).

Special Note
: *Thermocouples K17 and K18 are "extra", in the sense that they have been wired into the PXI module which does NOT have cold-junction compensation. This means that K17 and K18 are only useful for approximate temperature measurements (within a degree or two). For accurate temperature reading, you must use connectors K1..K16.*

RTDs
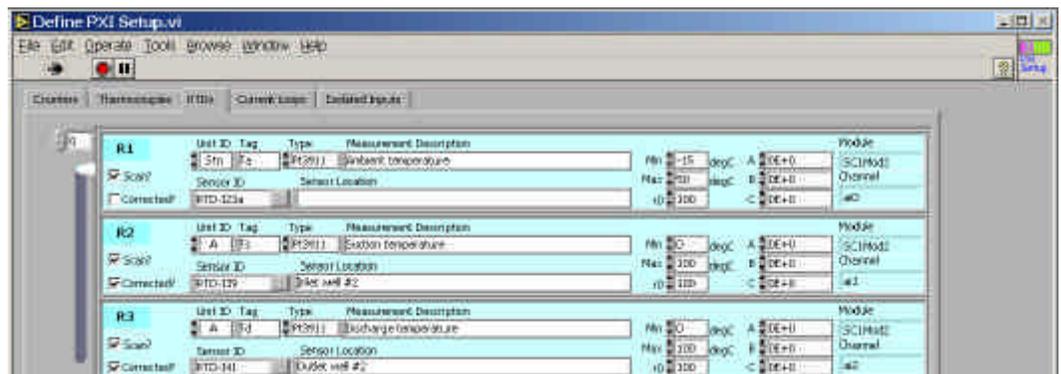
The RTD configuration is very similar to the thermocouple configuration. Some new or changed items are:

Type		A number of 4-wire RTD types are supported (see the following table). You can also specify the unique Callendar-Van Dusen coefficients for this particular sensor.
Min		The minimum allowable temperature, in the temperature units you specify to the right o the field. Note that you can right click on the units of measure field for help in specifying other units of measure.
Max		The maximum allowable temperature, in whatever temperature units you specify to the right o the field.
r0		The nominal impedance of the RTD at zero degrees Celcius. Typically 100 ohms.
A, B, C		Callendar-Van Dusen coefficients for this RTD. Leave these blank unless you have specified "Custom" as the RTD Type.

Platinum RTD Types

The following table lists the supported platinum RTD types and standards. Notice that there are some shared standards. The TCR and the Callendar-Van Dusen coefficients are more important than the standards[2].

Standards	TCR	Typical R_0 (O)	Coefficients	Notes
IEC-751 DIN 43760 BS 1904 ASTM-E1137 EN-60751	3851	100 O 1000 O	$A = 3.9083 \times 10^{-3}$ $B = -5.775 \times 10^{-7}$ $C = -4.183 \times 10^{-12}$	Most common RTDs
Low-cost vendor compliant RTD*	3750	1000 O	$A = 3.81 \times 10^{-3}$ $B = -6.02 \times 10^{-7}$ $C = -6.0 \times 10^{-12}$	Low-cost RTD
JISC 1604	3916	100 O	$A = 3.9739 \times 10^{-3}$ $B = -5.870 \times 10^{-7}$ $C = -4.4 \times 10^{-12}$	Used in primarily in Japan
US Industrial Standard D-100 American	3920	100 O	$A = 3.9787 \times 10^{-3}$ $B = -5.8686 \times 10^{-7}$ $C = -4.167 \times 10^{-12}$	Low-cost RTD
US Industrial Standard American	3911	100 O	$A = 3.9692 \times 10^{-3}$ $B = -5.8495 \times 10^{-7}$ $C = -4.233 \times 10^{-12}$	Low-cost RTD
ITS-90	3928	100 O	$A = 3.9888 \times 10^{-3}$ $B = -5.915 \times 10^{-7}$ $C = -3.85 \times 10^{-12}$	The definition of temperature

[2] This table is taken from the National Instruments LabVIEW documentation.

Current Loops
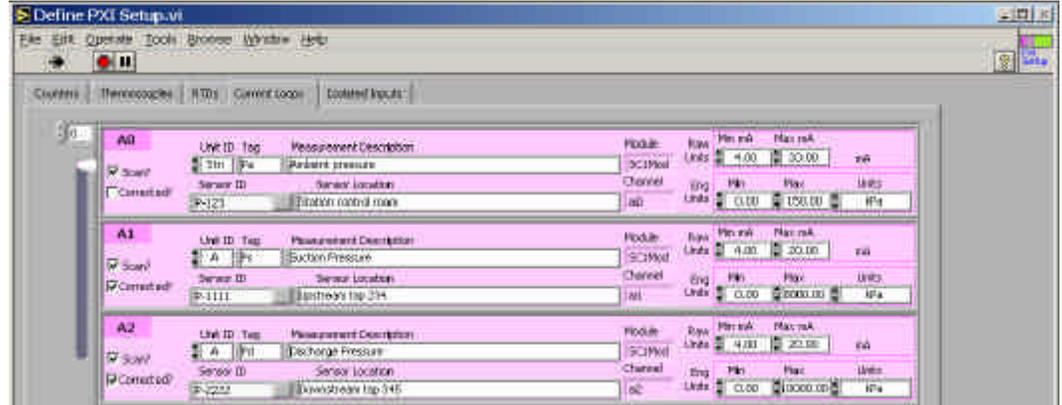
Almost anything than can be connected to a powered 4..20 mA current loop can be measured in this section (to hook up in series with an existing current loop, use the Isolated Input tab).

The raw signal shown in this panel is the current in miliamps produced by the transducer (typically 4..20 mA).

The SI unit list is longer than for the frequency/time inputs:
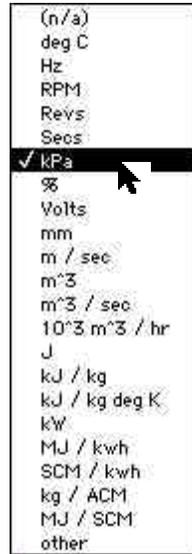
Some more things to consider:

Min .. Max   If your scaled measurement falls outside this range, it will be flagged
                              as being out of range.

Corrected Measurements

Measurements of speed, temperature, or pressure may also be "Corrected" to ambient conditions. You specify the ambient pressure and temperature in *Define Calculations.vi,* which can be reached by clicking "Calculations" in the main menu.

The corrections are made as follows:

$$d = \frac{P_a}{P_b}$$

$$q = \frac{T_a + 273.15}{T_b + 273.15}$$

$$P_c = \frac{P_f + P_a}{d} - P_b$$

$$T_c = \frac{T_f + 273.15}{q} - T_b$$

$$N_c = \frac{N}{q^j}$$

Where P, T, and N are pressures (kPa), temperatures (deg C) and speeds (RPM),
and subscripts a, b, f, and c are ambient, base, flowing, and corrected.

Isolated Inputs
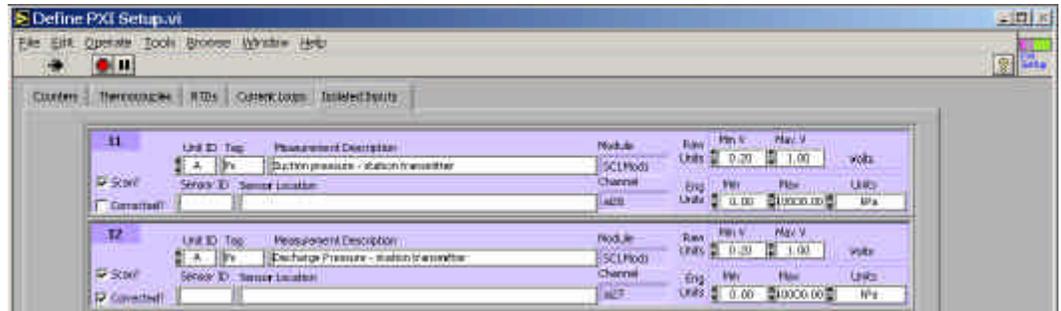
The TGN data conditioning system includes four isolated inputs, which are designed to be connected in series with a station current loop.

The current loop is converted to a voltage using a 50 ohm sampling resistor, and the voltage is isolated using and Analog Devices isolation amplifier.

The raw signal should always be in the range 0.2 .. 1.0 Volts. Other than that, these signals operate in the same way as the other current loop signals.

Derived Points

In addition to the signals you acquire from the PXI chassis, you can create four types of "Derived Points":

- Manual inputs
- Shared signals
- Averaged signals
- Sums and differences

You select which of these you wish to configure by selecting the appropriate tab

All four types are saved in the same file. By default, the system will propose *Station name.der* as the name for this file, but you can use any file name you wish. By default, only files with the extension .der are visible in the open and save dialog boxes.

Manual Inputs
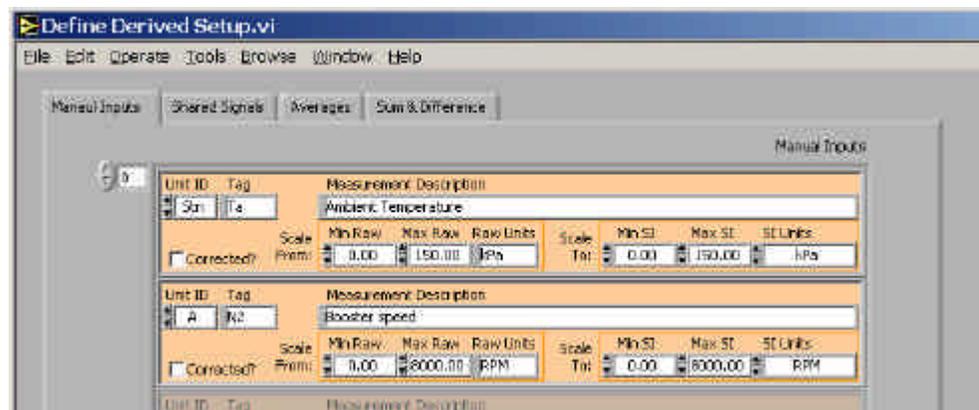
If you are unable to measure a signal, you can still input it manually.

If your measurements are for speeds, pressures, or temperatures, they can be corrected for ambient conditions.

Like measured signals, you can scale the values you enter to other engineering units.

Manual inputs are only requested when you add to the *Snapshot Buffer,* or on the first scan. You can use the most recent manual inputs, or change them (press Tab to move from item to item; Enter when you're done):

Tags	Values (raw)	Raw Units
1 :Ps	5000.0000	kPa
1 :Pd	6000.0000	kPa
1 :dPo1	42.0000	kPa
1 :Po1	5500.0000	kPa
1 :P2	6000.0000	kPa
1 :Pa	95.4500	kPa
1 :Ts	-5.0000	C
1 :Td	20.0000	C
1 :To1	-5.0000	C
1 :Ta	12.0000	C
1 :Tscrub	5.0000	C
1 :Tadj	3.0000	C
1 :N1	14000.0000	RPM
1 :N2	16500.0000	rpm

OK

Shared signals
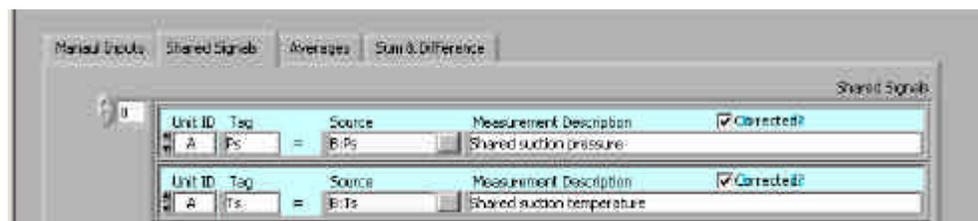
You can assign more than one tag to a signal. You can specify any PXI or manually input measurement.

Averaged signals
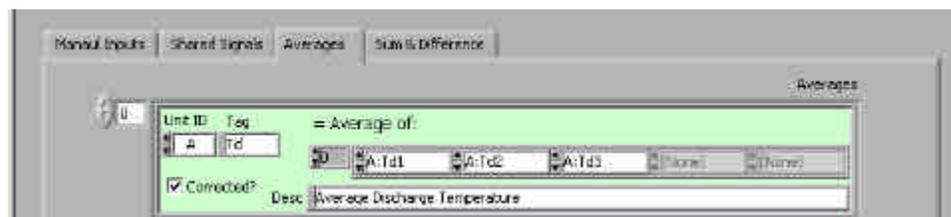
You can also average a number of signals together.

There is no limit to the number of signals you can average, and you can include signals more than once if you want to weight the average in their favour.

You can average any PXI, manual, or shared signals.

Sum and Differences
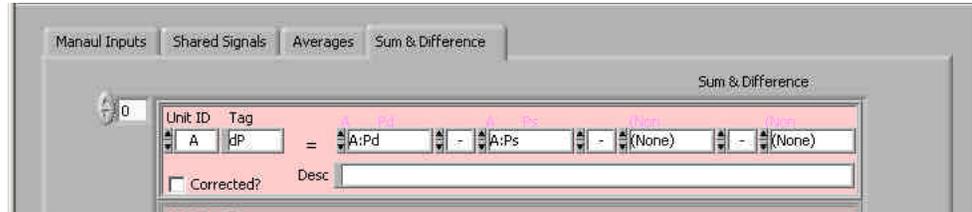
You can also apply addition and subtraction operators.

The pop-up menus show the PXI, manual, shared, and averaged signals, as well as all of the flow measurements (logically, you should define the flow devices, and then come back to define the Sum & Difference signals).

The unit of measure is determined by the first signal you specify. The system does **not** check the rest of the signals for compatible units.

If you check the "Corrected" checkbox and the unit of measure is compatible, the resultant measurement will be corrected for ambient conditions.

Flow Measurements

The Performance Testing software supports a number of flow measurements devices, each of which has a tag in the Define Flow Devices vi:

The flow measurement devices fall into two categories: those that rely on a pressure differential, and those that measure a positive displacement and generate an electrical pulse for each unit of gas which has passed through the device.

Since the windows for each meter type are quite similar, we'll just show one for each category.

In addition to the flow tag, two tags are automatically created for each flow device: tag.fpv, and tag.z (the supercompressibility and compressibility factors). These are calculated using either the 1985 or 1992 version of AGA-8, using the detailed calculation method.

To remove a flow device, give it an empty tag name. It will be removed when you click "Done".

The flow device setup is saved with a file extension of ".fld".

Orifice Plates
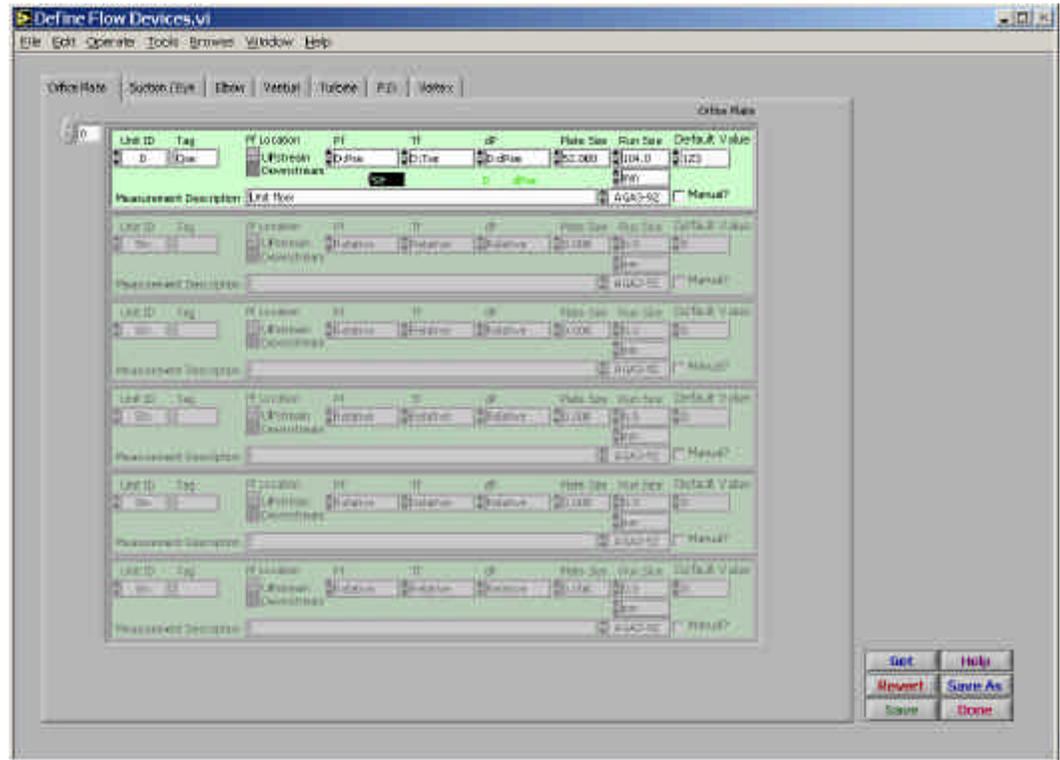
One of the most common (and most accurate) methods of flow measurement is the orifice plate. In a minor departure from the rest of this system, you can specify the run geometry in either inches or millimeters.

The flow calculation can be performed using either the 1985 or 1992 version of the AGA-8 and AGA-3 calculations (you specify the AGA-3 calculation; the corresponding AGA-8 calculation is also performed). Of the two calculations, AGA-8 is by far the most time-consuming.

Although the AGA-8 calculations can be made using either the "detailed" method, or one of two "gross characterization" methods, the Performance Testing software always selects the detailed calculation method. The time required for this calculation is dependent on the number of gas components which were defined in the Station setup.

It's assumed that the calibration you defined for dP already includes any necessary static pressure and offset compensation, as well as any necessary temperature compensation. There is no provision for dual-ranged dP signals.

Suction / Eye and Elbow Meters

These flow measurement methods are similar in principal to orifice plate measurement, but are usually only valid over a relatively small flow range.

In practice, a flow run defined using one of these techniques is compared with an orifice plate on the same run. Using the orifice plate as a reference flow device, a coefficient is experimentally derived for the suction / eye or elbow meter.

The Performance Testing software can calculate these coefficients — to do so, click "Meter Coefs" in the main menu (don't do this until you've defined all of the flow devices):
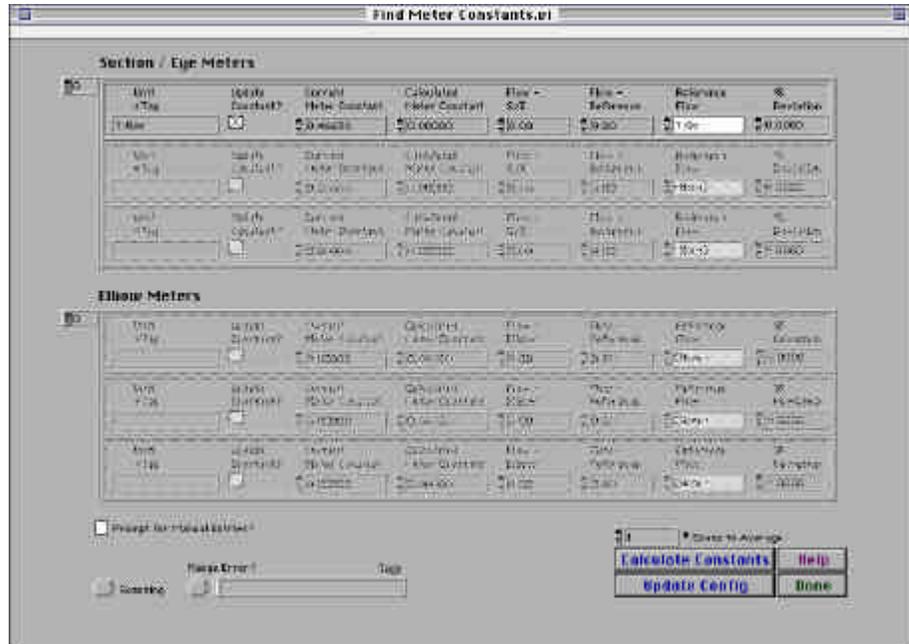

All of the Suction / Eye and Elbow meters you have defined appear in this window. Check "Update Constant?" for each flow measurement for which you wish to calculate the meter coefficient.

Make sure that the reference flow device is correct for each of these flow measurements, and click "Calculate Constants".

This VI will average a number of "Normal Scans" together, and use the calculated flows to calculate a new meter coefficient (remember, only for the flows where you checked "Update Constant").

If you wish to update your configuration with the new coefficients, click "Update Config" button to view the flow device configuration screen. The meter coefficient will be automatically updated for each meter for which you checked "Update Constant?". Click "Save" to save your changes, and "Done" to return to this screen.

Vortex and Turbine flow meters
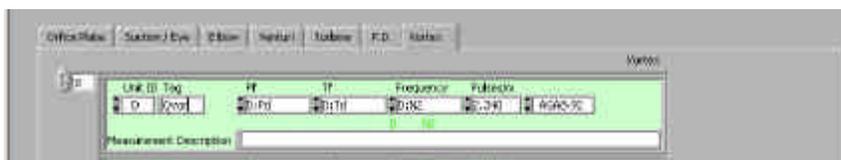
Both vortex and turbine meters require a frequency measurement, which is normally supplied by one of the counter / timer inputs to the PXI.

To convert the actual flow measured by these meters to standard flow, the AGA-7 calculation is used (the 1985 version is always used, as there is no 1992 version).

P.D. meters

Positive Displacement meters operate in the same way as vortex and turbine meters, except that a time measurement, rather than a frequency measurement, is required.

Venturi Meters

Although you can enter information for a Venturi meter, the Performance Testing system doesn't calculate the flow (you still get the AGA-8 compressibility and FPV values).

Define Calculations

In addition to flow calculations, there are two other main groups of calculations: Equation of state (compression thermodynamics), and ASCV capacity calculations.

To the right of the screen, are the Station Conditions, which you always need to specify:
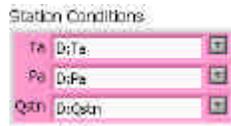
You can either type in the tag names, or select them from a pop-up by clicking the arrow to the right of the field. If you type the tag name in, the system will try to auto-complete the tag name, so you only need to type in enough characters to make the tag unique. Note that you can't type in the name of a non-existent tag.

Compression Calculations
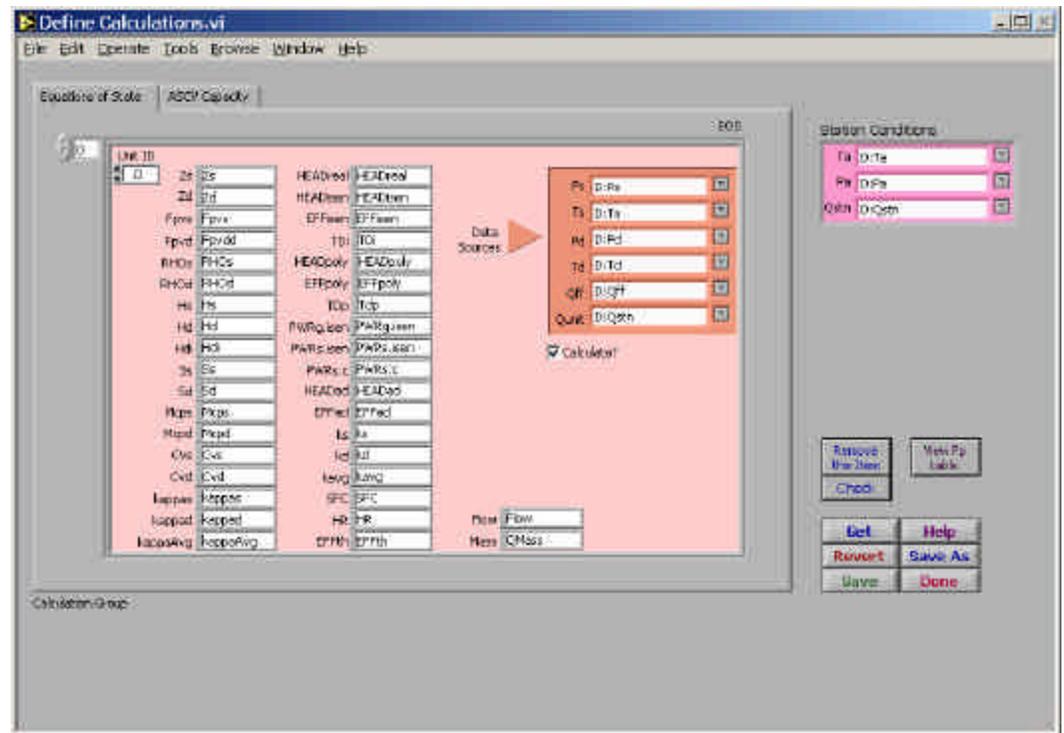
When you create a new calculation, you need to tell the system where to find the inputs to the calculation:
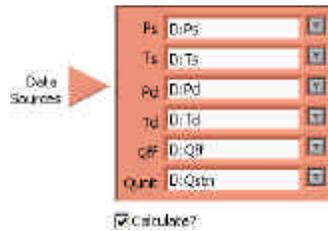
As with the station conditions, you can either type in the name of the tag, or click the little arrow to select from a pop-up list. For your convenience, only tags with a compatible Unit of Measure will appear in the pop-up list.

If you don't set the "Calculate?" checkbox, the compression calculations will not be performed. Remember that you can request as many calculations as you want, .

Each calculation produces a large number of results. Each result is assigned to a tag. The left-hand portion of the panel defines all of the tags which will be produced by the calculation. They are assigned default values, which you can change if you wish.

You can perform as many equation of state calculations as you want. Each entry in the Equation of State array corresponds to one set of calculations. You can skip these calculations for some compressor units, or perform them more than one for the same unit. If you perform the calculations for the same unit more than once, remember to give the results unique names.

Please note that you need to specify the Station Conditions in the upper portion of the window (they apply to all of the Equation of State and ACSV calculations).

By default calculation definitions are saved in a file named *Station*.equ.

Checking for missing measurements

If you click "Check", the system will check for missing measurements, and display a message itemizing any that are missing.

ASCV Capacity
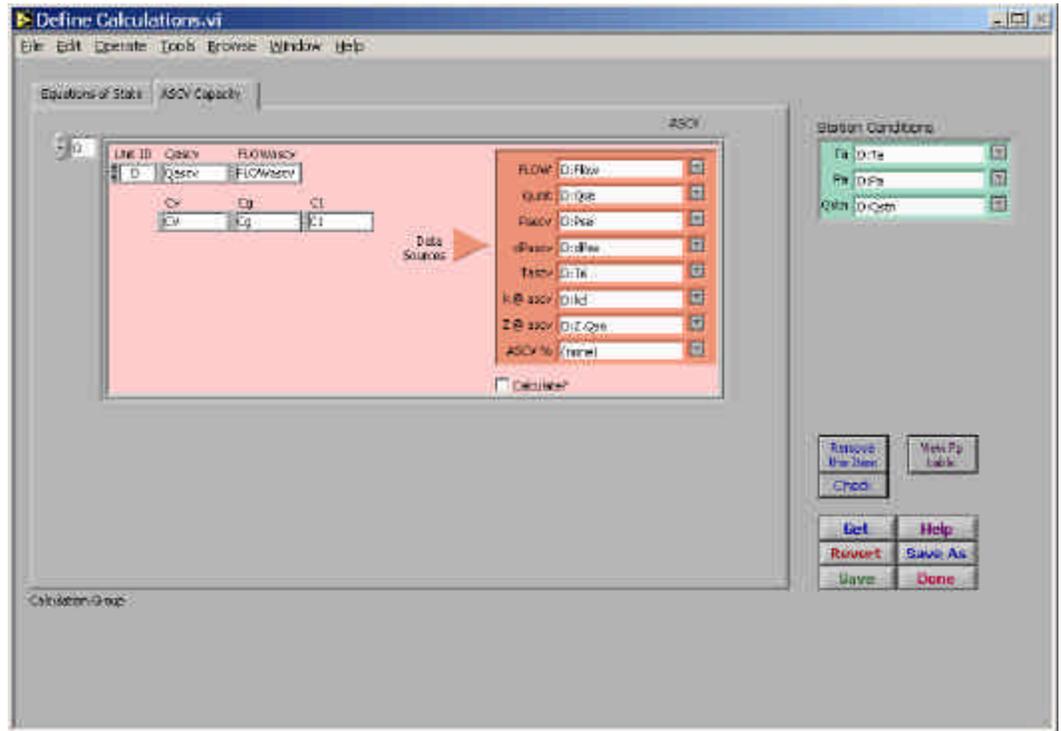
You can also calculate the anti-surge control valve capacity for any number of compressor units.

Configuring the ASCV capacity is similar to configuring the Equation of State calculations. The k and Z inputs normally would be the kd and Zd calculated values for this unit.

You can look up Fp tables by clicking "View Fp Table":

## Typical Piping Geometry Factors

Valve Size (in)	d/D				
	0.50	**0.60**	**0.70**	**0.80**	**0.90**
Mark 1 & 2, Class 150-1500					
1/2, 3/4	0.91	0.93	0.96	0.98	0.99
1..6	0.94	0.95	0.97	0.98	1.00
8..24	0.96	0.97	0.98	0.99	1.00
30..48	0.92	0.94	0.96	0.99	1.00
Class 2500					
1/2..16	0.98	0.98	0.99	0.99	1.00
Valdisk					
3	0.84	0.87	0.91	0.95	0.99
4, 6	0.80	0.84	0.88	0.94	0.98
8..12	0.77	0.82	0.87	0.93	0.98
14..24	0.70	0.75	0.82	0.90	0.97

Fp, Expander Only Full Area Trim

Valve	d/D				
Size (in)	**0.50**	**0.60**	**0.70**	**0.80**	**0.90**
Mark 1 & 2, Class 150-1500					
1/2, 3/4	1.05	1.06	10.70	10.60	1.05
1..6	1.03	1.04	1.04	1.04	1.03
8..24	1.02	1.03	1.03	1.03	1.02
30..48	1.04	1.05	1.06	1.05	1.03
Class 2500					
1/2..16	1.01	1.01	1.01	1.01	1.01
Valdisk					
3	1.11	1.14	1.15	1.14	1.09
4, 6	1.16	1.21	1.24	1.21	1.12
8..12	1.20	1.27	1.31	1.27	1.16
14..24	1.36	1.52	1.62	1.52	1.28

Data Management

When you've completed the setup steps, or if you wish to resume a test, click "Test Prep." in the main menu:
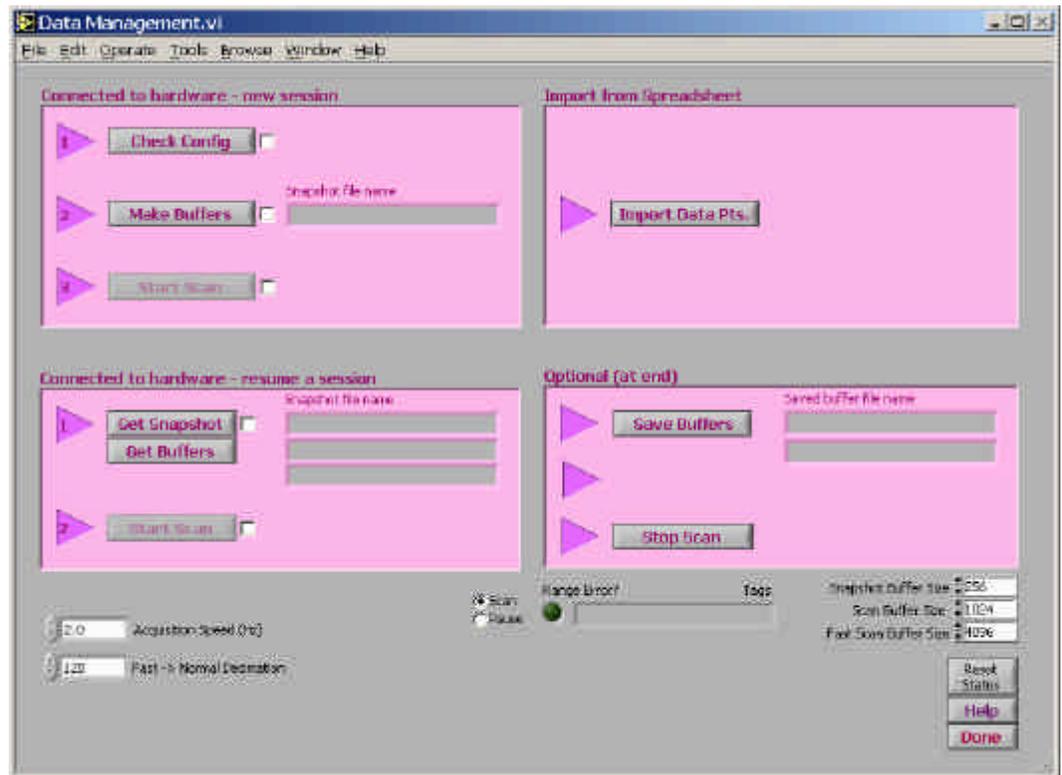
There are four coloured sections on this window, corresponding to different things you might wish to do:

A new Session

If this is a new test, and you've gone through the setup steps, use the "Connector to hardware — new session" section.

"Check Config" makes sure that you've set up the flow and equation of state calculations correctly. If there's an issue, you'll see a warning message:

When you click "Make Buffers", the system will create in-memory buffers:

The Performance Testing system uses three main types of buffers to hold signals acquired from the PXI chassis and calculated values: *Snapshot Buffer, Normal Scan Buffer,* and *Fast Scan Buffer.*

Each of these circular buffers can be a different size, selectable from the pop-up menus at the bottom of the window, or in "Preferences".

Snapshot Buffer Size 256
Scan Buffer Size 1024
Fast Scan Buffer Size 4096
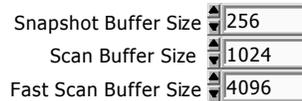
It's important to define your test configuration before you create a buffer set, because each buffer "knows" how many channels it can hold. If you change the number of channels (by adding or removing tags from the scan list), you need to create a new buffer set.

Snapshots are always saved to disk immediately, so you are asked to create a file to hold Snapshots as soon as the buffer is created. Normal and Fast Scan buffers are only created in memory.

We suggest you make the Snapshot Buffer larger than you think would be necessary. If you exceed the number of allotted points, the in-memory Snapshot Buffer will re-use the earlier points (although the file on disk will still contain all of the points).

Sampling rates

Two important setting to check before starting the scan are:

2.0 Acquisition Speed (Hz)

120 Fast -> Normal Decimation
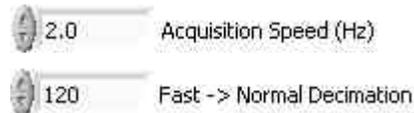
The acquisition speed is the rate at which the PXI chassis is sampled. Due to the calculations which are performed on the signals, a rate of about 2 Hz is suggested. Signals are acquired into the "Fast" buffer. Every so often, a certain number of "Fast" signals are averaged together to form a new "Normal" signal set.

If you acquire at 2 Hz, and use a Fast->Normal decimation factor of 120, a new "Normal" point will be created every minute.

After you've created a set of buffers, click "Start Scan" to start data acquisition. The scan indicator moves to "Scan":

● Scan
● Pause

A new window is added to the task bar, called "Acquire PXI.vi". The window is normally minimized, but you can view it to monitor the signals as they are acquired:
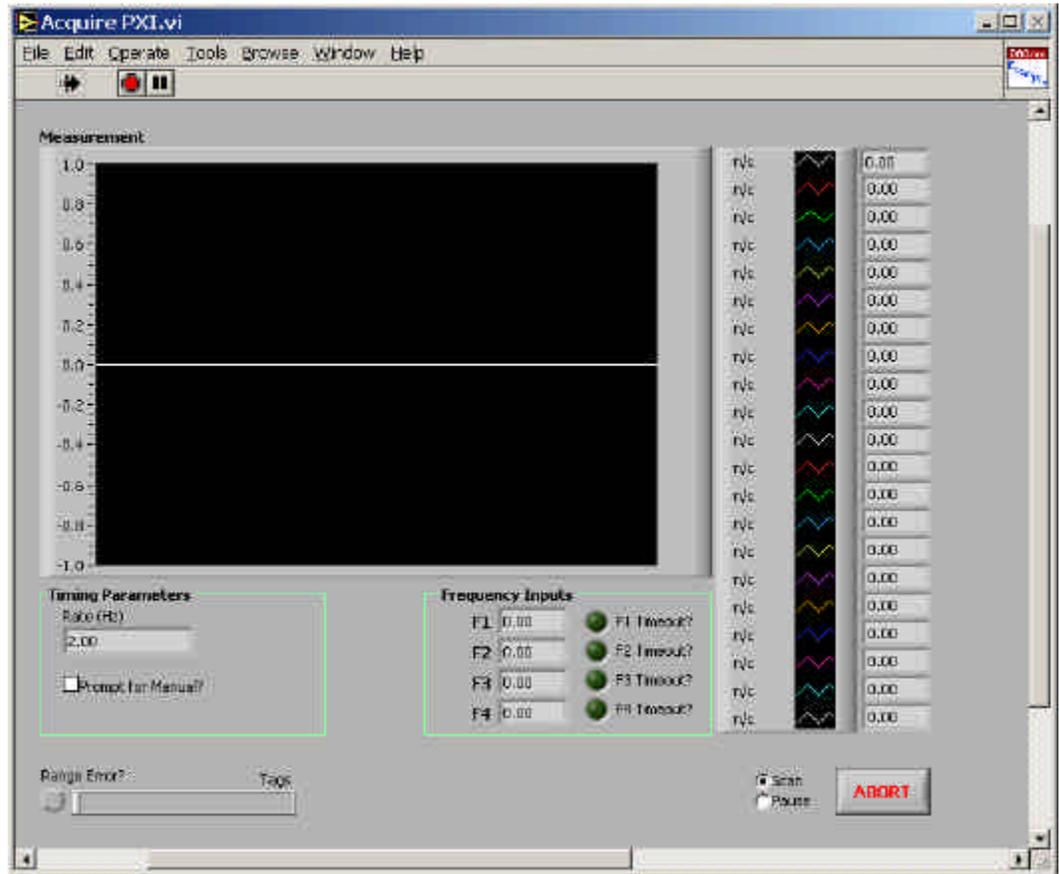
If a signal goes out of the range limits you defined in "PXI Setup", the red LED illuminates, and the out-of-range tags are displayed:

Range Error?   Tags

 M9:N3,Stn:Pa

In this example, the N3 signal from unit M9 is out of range, as is the Station-level signal Pa.

Resuming a previous session

If you are connected to the PXI system, and wish to resume a previous session, you would work with the items in "Connected to hardware — resume a session":

1. **Get Snapshot**

Get a previously saved Snapshot file. The entire configuration is saved inside the Snapshot file, so when you read the Snapshot file, it also recreates all of the configuration screens.

Get Buffers

The background buffers (Normal and Fast Scan buffers) can be saved to disk, or restored from disk to memory. If you restore a Normal Scan buffer, it can be restored as a Snapshot Buffer, to allow you to recalculate its contents.

Note: Each of the buffers also contains a complete configuration — when you retrieve a buffer, you also retrieve the configuration which was in place *at the time the buffer was created*.

2. **Start Scan**

Operates the same as in the previous section.

Import from Snapshot

You can also import data points from a spreadsheet file. This section simply duplicates "Import Pts." on the main menu.

Please see that section of the User's Guide for full details.

Finishing Steps

If you wish, you can save the in-memory buffers to disk at the end of a session. At a later time, you can restore them back into memory, if you are resuming the data acquisition session.

You can also click "Stop Scan" to stop acquiring data from the PXI system. It's a good idea to stop the scan before disconnecting the PXI system.

Important Warning

Remember that changing the configuration can affect the number of tags in the scan list, so you must not do so after you have created buffers!

Monitor All Analogs

This VI lets you monitor every analog connector on the PXI chassis. Each connector is shown in a box like this:
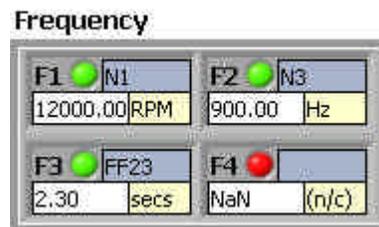
The name of the connector is in the upper left. Next to it is an indicator which is green if the connector is part of the scan, and red otherwise. The signal tag is next, with the current signal level and units of measure on the bottom. They layout mimics the connector panel of the TGN self-contained PXI chassis.

You can monitor each signal in SI units, or in your currently preferred units of measure:

You can also change your unit of measure preferences:

Troubleshooting

- Frequencies missing:

  The frequency / time inputs are the most likely to present problems. The most common problem is that the input signal is not a true TTL signal. You may need to introduce additional signal conditioning, such as a Schmitt trigger. Also, if you are not using external isolation, you should remember that the frequency inputs are single-ended, and may be susceptible to differences between the ground plane of the PXI system, and the ground reference for your signal.

  Also, the frequency inputs need to have minimum and maximum expected ranges. You may not have allowed enough time for the frequency to be measured, or it may lie outside the specified range (this range is different from the range used to signal "Out of Range".

- Can't see tags:

  Only a limited space is available on this panel to display tag strings. If the string is too long

to fit, it may appear blank. This does not affect the operation of the system, and is not a cause for concern.

Note   Depending on your preferences, signals will either be set to be set to "NaN" (Not a Number) or to a very high number if they go out of range.

Gas Generator Schematic

You can also monitor the signals for a compressor unit on a "schematic" display like this:
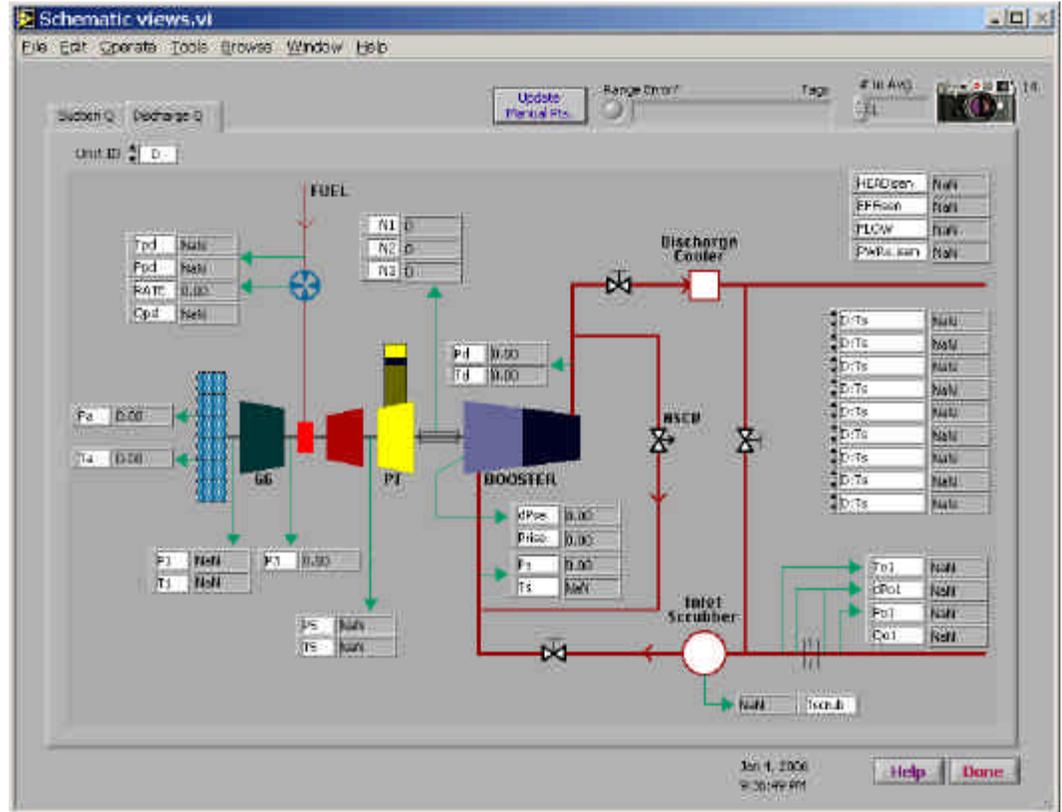
The monitoring points are preset to common tag names, but you can change any of them. If you are monitoring a signal with a tag from a different unit than the one you are viewing, you need to quality the tag with the correct unit name.

You can quickly switch between units:

Unit ID   1

There are two versions of the schematic: one for units which measure flow at the suction side of the unit, and one for units which measure flow at the discharge side.

If you want to monitor additional points, there is a block of pop-up menus in the right-hand portion of the screen:

(None)	NaN
(None)	NaN
(None)	NaN
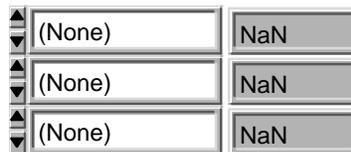
These points do not change when you switch units.

On this panel, as with most of the operating panels, you can elect to capture Snapshots. Click the camera icon to average one or more "Fast Scan" points into a new Snapshot point. The number to the right of the camera icon is the number of Snapshots you have already acquired:

Strip Charts

Using Strip Charts

Either of the two strip charts on this window allows you to view up to half a dozen signals in a buffer. These can be selected in any order from the pop-up menus located to the left of the strip chart:
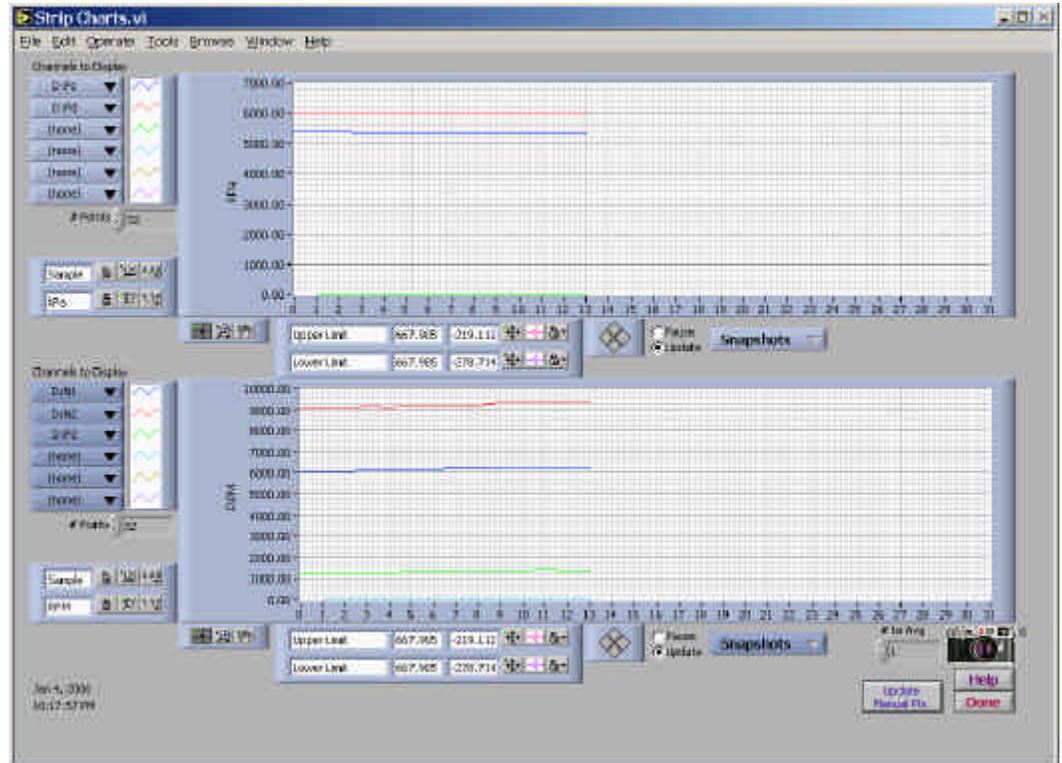
Each of the strip charts can be set to view Snapshots, Fast, or Normal points:
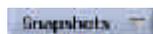
The number of points to be displayed is determined by the # Points setting.

You can also place cursors on the plots to measure signals. If you do, you may wish to pause the display, so that the graph doesn't change while you are using the cursors:

If you want, you can control how the plots are displayed by clicking one of the coloured lines to the right of the pop-up menus (quite a number of options are available):
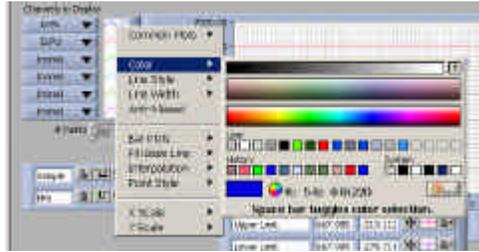
Please note that the time axis of the Snapshot strip charts will not be uniform, because points are acquired only when you ask for them.

On this panel, as with most of the operating panels, you can elect to capture Snapshots. Click the camera icon to average one or more "Fast Scan" points into a new Snapshot point. The number to the right of the camera icon is the number of Snapshots you have already acquired:

Operating Point

The Operating Point display lets you view the current operating point and snapshots on top of a couple of previously created wheel maps[3] or X-Y graphs[4].

By default, the system will attempt to display the most recently displayed wheel map and X-Y graph. If you haven't already displayed any of these graphs, you select them using the "Setup" tab.

It's OK to specify the same graph on both the right-hand and left-hand side. If you want, you can zoom in on an interesting part of the graph on one side, and display the whole graph on the other side[5].

Overlaid on these are two more plots:

- An X-Y graph of all the points in the current Snapshot Buffer (blue crosses).

- The current operating point (from either the Slow (Normal) Scan or Fast Scan buffer), plus a selectable number of previous operating points (red squares). The current operating point is highlighted with a crosshair cursor.

Associated with each wheel map or X-Y graph are two pop-up menus from which you can select the signals used to display the operating point. The system initially attempts to set them to sensible values, but you can change them at any time.

The number of previous operating points to display is selectable:

<div align="center"># Points ⬍ | 32 |</div>

You can also display strip charts — on them this number also sets the number of points to display in the strip charts.

Note   To clear the X-Y graph on the Wheel Map vi, click "Get Saved X-Y Graph", and then "Cancel" in the file dialog.

---

[3]   To find out how to create Wheel Maps, please refer to "Wheel Maps", later in this Guide.

[4]   Creating X-Y Graphs is discussed in "X-Y Graph", later in this Guide.

[5]   This is particularly handy during surge testing - you can zoom into a wheel map on one side, and display the entire map on the other side.

Create Wheel Map
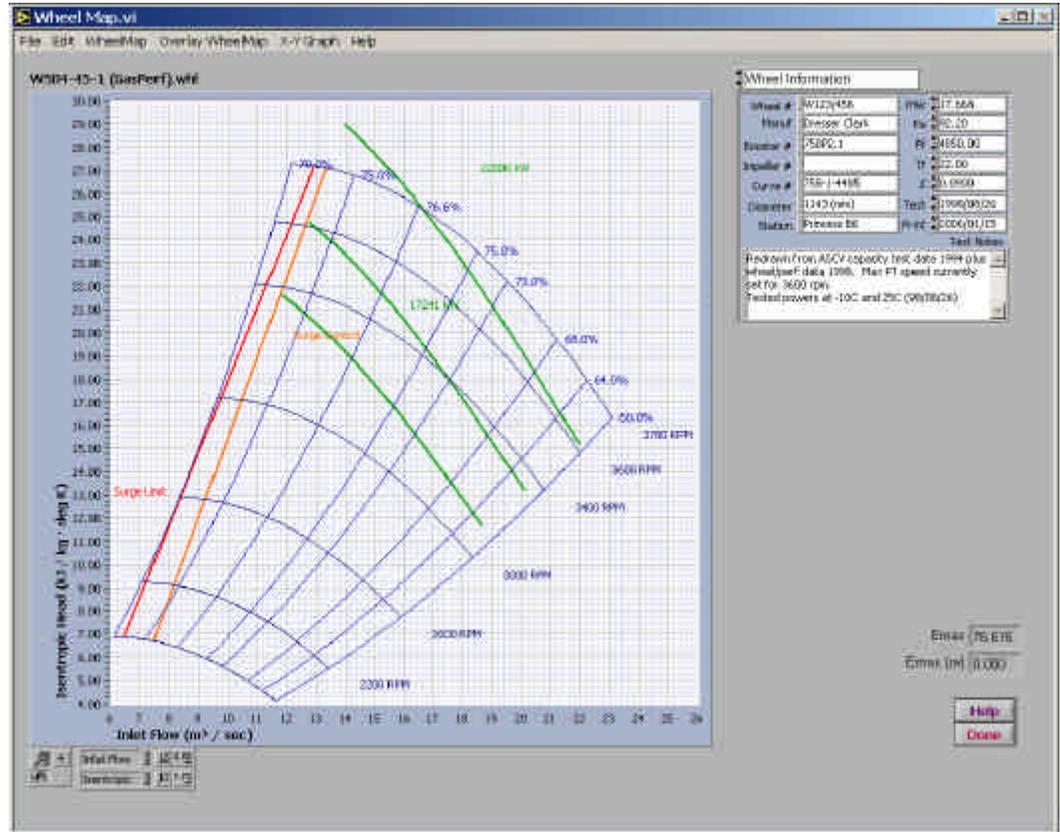
This VI ("Virtual Instrument") lets you create wheel maps from Cheesman & Wegmann coefficients, or from data points. You can display any number of speed, efficiency and power lines in your choice of colour and line style. You can display two wheel maps at a time.

The graph on the left of the screen contains the wheel map (you can overlay a second wheel map, and compare them).

There are a large number of controls for the wheel map, divided into several categories. You select the category to display from the pop-up menu in the top right-hand corner of the screen. The pop-up menu offers several choices:

Each choice causes a different group of controls to be displayed in the right-hand portion of the screen. Here are the categories:

Data Sources	Get the data points used to create a wheel map, fit a set of polynomial coefficients to the data points, and save wheel map files.
Coefficients	Cheesman coefficients used to produce speed and efficiency lines, plus coefficients for the control and surge lines.
Coefficients (Overlay)	Used to produce speed, efficiency, and control lines on the overlay wheel map.
Coefficients (Power)	Used to produce gas compression power lines on the wheel maps.
Wheel Map Range	Specify the speed, efficiency, and power lines.
Move Labels	Fine-tune the location of the label for each line.
Select Colours	Specify the colour and line style of each line.
X-Y Graph Legend	Clears out space so you can see the legend of the X-Y graph (if you are displaying one).
Wheel Information	Display information about the wheel: you would typically print wheel maps using this option.

This VI lets you create wheel maps from Cheesman & Wegmann coefficients, or from data points. You can display any number of speed, efficiency and power lines in your choice of colour and line style. You can display two wheel maps at a time.

There are a large number of controls for this VI. To manage them, they are divided into several groups, only one of which is visible at a time:

Data Sources

Most of the controls in this section are duplicated — one set for the main wheel map, and one for the overlay wheel map (if any).

Any of the operations you can perform on the main wheel map can also be performed on the overlay map.

Wheel Map Data	Overlay Data
Get from Snapshots	Get from Snapshots
Get from Recalc	Get from Recalc
Get saved Wheel Map	Get saved Wheel Map
Fit Coefs to data	Fit Coefs to data
Fit Coefs to surge data	Fit Coefs to surge data
Save as new Wheel Map	Save as new Wheel Map
Save this Wheel Map	Save this Wheel Map

Wheel file name	Overlay file name
Vetch.whl	

Get from Snapshots

You can extract a set of data points from the current Snapshot Buffer:

Import Wheel from Snapshot.vi

BWR Efficiency
1 :EFFbwr

Adiabatic Eff
(none)

BWR Head
1 :HEADbwr

Adiabatic Head
1 :HEADad

Speed
1 :N1

Flow
1 :FLOW

Replace

Cancel
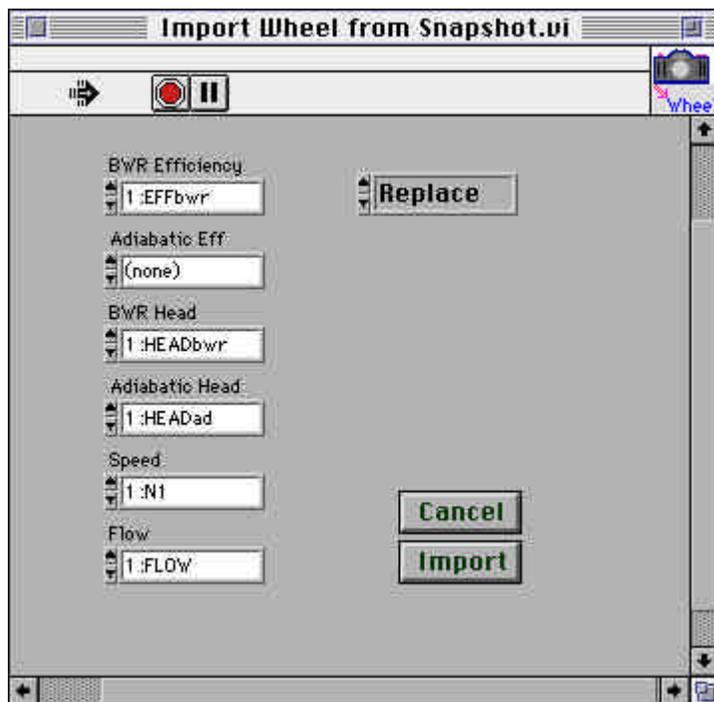
Import

The import function attempts to locate appropriate tags (for the first unit in the Station configuration), but you can select the measurement you want from the pop-up menus.

You can either create a new data set (the default, selected using "Replace" in the pop-up menu), or else append the points from the Snapshot Buffer to the end of the current set of data points.

After selecting the measurements you want, the "Fit Coefs to Data" window will be displayed.[6]

You can also import measurements from a Recalc file[7]:
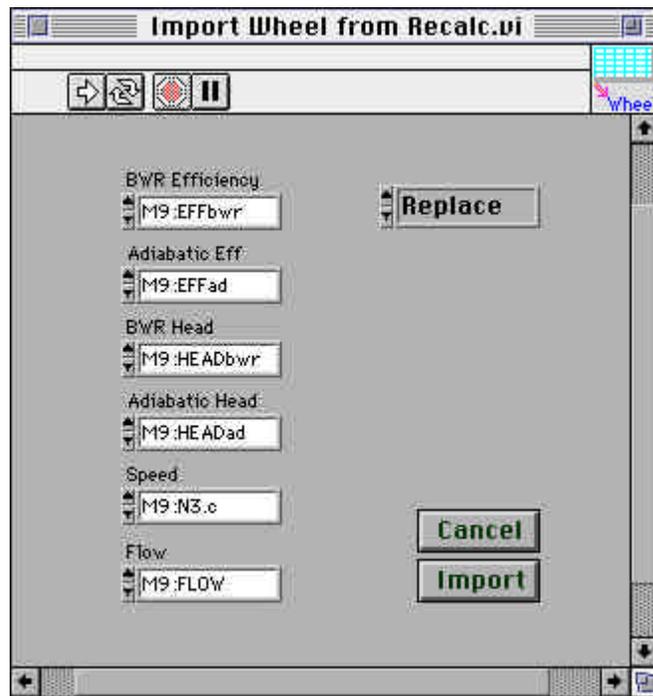
This function works just like the Import Wheel from Snapshot VI.

Get Saved Wheel Map

You can select any previously saved wheel map (the file name normally ends in ".whl".

Fit Coefs to data

This VI lets you create Cheesman and Wegmann coefficients from the current set of data points. You can manually add data points, and control which data points are used to compute the coefficients:

---

[6]   This function is described below in "Fit Coefs to Data".

[7]   The Recalc file is just a tab-delimited text file, which gives you an easy way to import data sets from other applications.
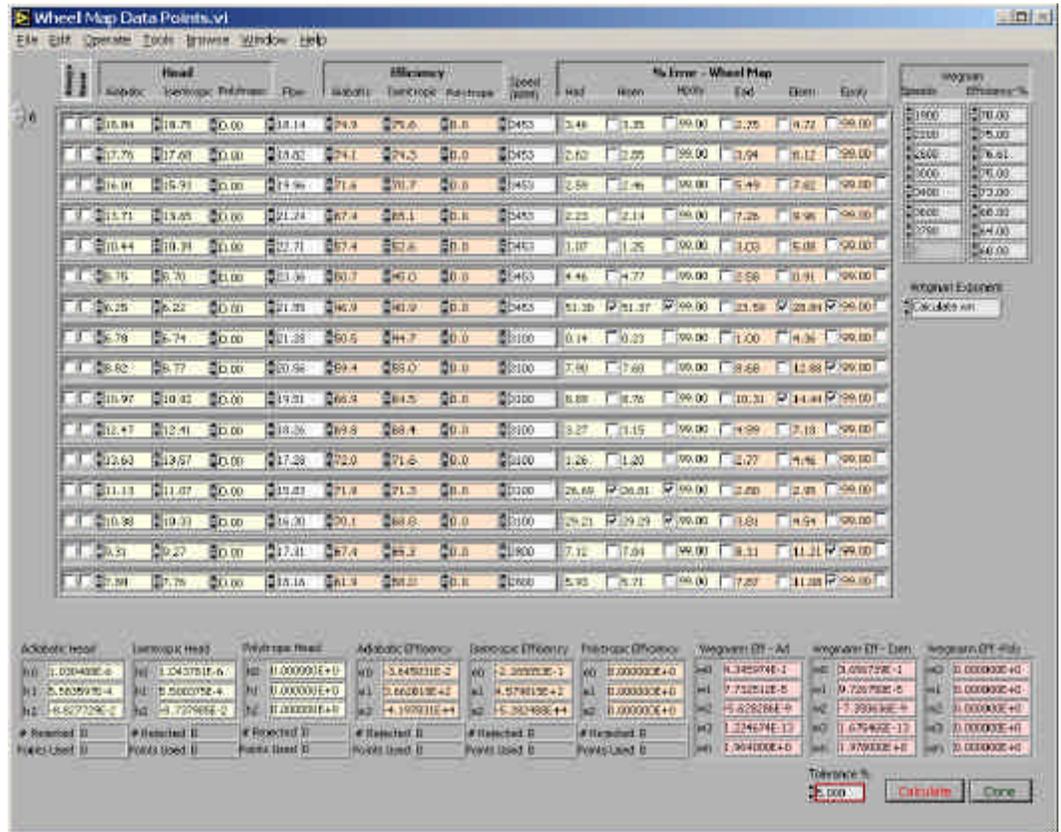
The main control in this window is a large array. Each row in this array describes a measured data point for a wheel map:
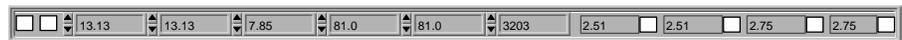
You can elect to always use a particular data point, never use it, or to use the data point if it falls within "Tolerance %" of the fitted polynomial.

If the "Always" box has been checked, the point will never be removed from the set of points used to fit the head and efficiency curves.

If "Never" is checked, the point will never be used, no matter how well it fits.

Cheesman Coefficients   The process is iterative — after a least-squares fit, points which fall more than "Tolerance %" of the fitted polynomial are removed, and the least-squares fit is performed again until no more points are removed.

If the number of points removed falls below three, a warning message is displayed.

Along with the coefficients for each of the Cheesman polynomials, the system tells you how many points were remove, and how many were used:

BWR Head		Adiabatic Efficiency	
h0	5.68078E-7	e0	4.54032E-2
h1	9.33397E-4	e1	6.79312E+2
h2	-2.56877E-1	e2	-1.46066E+5

# Rejected	2	# Rejected	0
Points Used	68	Points Used	70

Wegmann Coefficients

Although it is possible (and correct) to simply fit Wegmann polynomial coefficients to the measured data, there is a fundamental problem with Wegmann's polynomial for efficiency:

$$Efficiency = w_0 + w_1\left(\frac{Head \cdot 10^5}{Flow^2}\right) + w_2\left(\frac{Head \cdot 10^5}{Flow^2}\right)^2 + w_3\left(\frac{Head \cdot 10^5}{Flow^2}\right)^3 \text{ Quite}$$

simply, efficiency increases without bounds as the flow rate decreases.

We know that efficiency actually drops near the left-hand side of the wheel map (as flows decrease). In the simplified world of Cheesman, the maximum efficiency is a second order curve, and efficiency decreases to either side of that curve.

When we actually measure points for a wheel map, it is somewhat unlikely that we will measure many points to the left of maximum efficiency, as there is a danger of surge conditions existing in this region of the wheel map.

This means the constant power lines based on Wegmann efficiency will tend to rise dramatically to the left of the maximum efficiency curve, even if we cap the efficiency at 100%

A solution of sorts is to use more point to the left of maximum efficiency when we are creating the Wegmann coefficients.

We actually do this by using the points on all of the speed lines, within the total range of efficiencies over which the wheel map is drawn (using Cheesman coefficients).

If you already have a wheel map drawn, the speeds and efficiency range are automatically transferred to this VI:

Wegman	
Speeds	Efficiency %
3200	75.00
3500	80.00
4000	82.00
4500	83.52
5000	82.00
5500	75.00
	70.00
	65.00

If you wish to change these lines, you will only affect the range of the wheel map which is used to create the Wegmann coefficients. Note that you must supply at least two speeds, and three efficiencies (left-hand side minimum, maximum, and right-hand side minimum).

By default, 50 points are taken on each speed line, uniformly distributed across the efficiency range you specify.

This will tend to force the Wegmann polynomial to be somewhat constrained over the range of the wheel map.

Calculating w_n

In Wegmann's polynomial:

$$Efficiency = w_0 + w_1\left(\frac{Head \cdot 10^5}{Flow^2}\right) + w_2\left(\frac{Head \cdot 10^5}{Flow^2}\right)^2 + w_3\left(\frac{Head \cdot 10^5}{Flow^2}\right)^3 \text{the Flow}^2$$

terms fit best if the power is slightly less than two.

GasPerf can calculate this exponent, w_n in two ways: setting it to 2.0, or actually determining the exponent which makes the polynomial fit the best:

√ Calculate wn
Use wn = 2
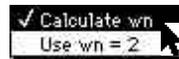
Here is an example calculation, showing the effect of w_n on the total mean-squared error in the polynomial fit (the example is for wheel W127-36-1, installed at Nova's Vetch Lake compressor station). W_n is on the X-axis, and error is on the y-axis.

Effect of wn on mean-squared-error
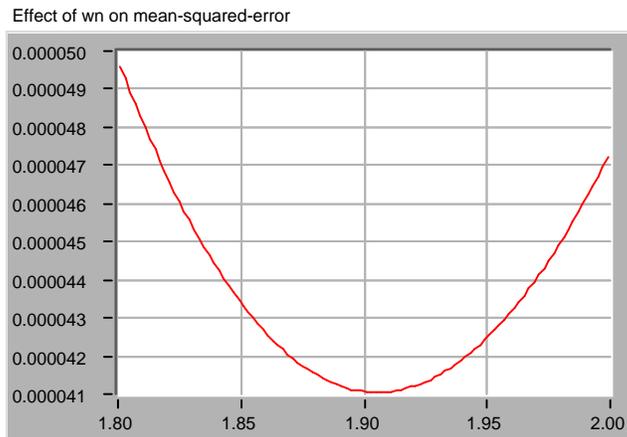
As you can see, the effect is somewhat subtle, but is still significant. The calculation of w_n can take two or three seconds.

Fit Coefs to surge data

The wheel map includes two special lines: the *Surge Limit line,* and the *Surge Control line.*
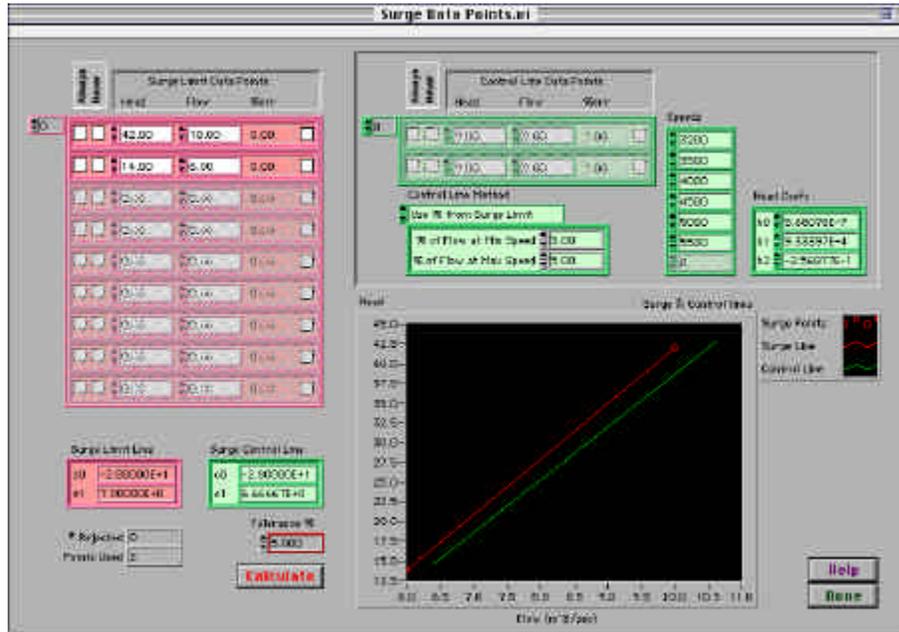
You can fit the Surge Limit line coefficients to a set of points which you manually enter:
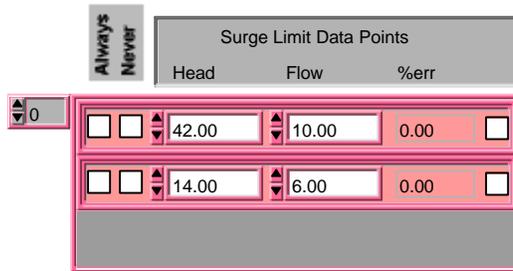
You only need to enter two points to define a line, but you can enter as many as you wish. This VI will perform a least-squares fit to create a first-order polynomial (line) which best fits the points you enter.

You can mark points as "Always", meaning that they are always used, no matter how badly they fit, and "Never", meaning that they are never used, no matter how well they fit.

Remember, you need to click "Calculate" to calculate and plot both the Surge Limit line and the Surge Control line:
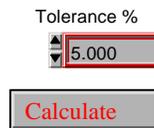
Like the Surge Limit line, you can fit the Surge Control line by entering a set of data points.
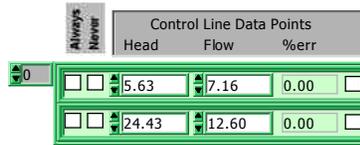
Alternatively, you can offset the Surge Control line by a percentage of flow from the Surge Limit line:
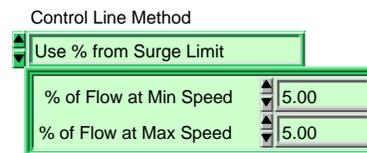
Note that if you use this method, you MUST define a minimum and maximum speed, as well as the Cheesman coefficients for Head (they are used to calculate the flow values at the upper and lower end of the Surge Limit line). If the speed lines and head coefficients have been defined in the main wheel map VI, they are automatically filled in for you.

Save as / Save

When you save a wheel map, you save everything needed to create the wheel map:

- All the data points for the wheel map.
- The data points for the surge limit and surge control lines, and the method used to draw the surge control line.
- The Cheesman and Wegmann coefficients.
- Coefficients for the surge limit and control lines.
- The speeds, efficiencies, and power lines to be drawn.
- The colours and line styles of each line.
- The X and Y range of the wheel map.
- The location of each label.

The actual wheel map plot is not saved: it's redrawn whenever it's needed.

Wheel file name

Shows the name of the current wheel file is displayed, along with the name of the overlay wheel map.

Wheel Map report

You can also create a text report showing the wheel map configuration:
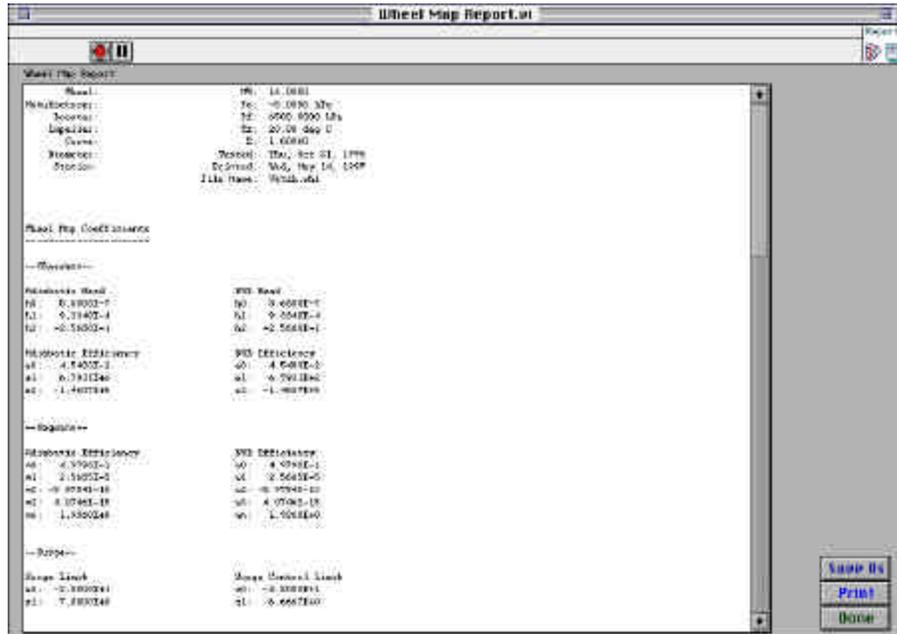
You can view the report (the scroll bar is "live"), print it out, or save it as a text file.

Cheesman Coefficients

The Cheesman polynomials are an attempt to simplify wheel map characterization by using *fan law* to reduce the independent variable of a polynomial to the term $\left(\dfrac{Flow}{Speed}\right)$.

$$Head = Speed^2\left(H_0 + H_1\left(\frac{Flow}{Speed}\right) + H_2\left(\frac{Flow}{Speed}\right)^2\right)$$

Where

(H_0, H_1, H_2) are the Cheesman Coefficients

$$Efficiency = e_0 + e_1\left(\frac{Flow}{Speed}\right) + e_2\left(\frac{Flow}{Speed}\right)^2$$

Where

(e_0, e_1, e_2) are the Cheesman Coefficients

You can view and modify the Cheesman coefficients for both adiabatic and isentropic head and efficiency. Nova currently uses second-order Cheesman coefficients, although third-order coefficients are known to be more accurate.

In most cases, you'll use the "Fit Coefs to data" option under Data Sources to create new coefficients from a set of data points.

Adiabatic Efficiency		Adiabatic Head	
e0	4.54032E-2	h0	5.68078E-7
e1	6.79312E+2	h1	9.33397E-4
e2	-1.46066E+5	h2	-2.56877E-1

BWR Efficiency		BWR Head	
e0	4.54032E-2	h0	5.68078E-7
e1	6.79312E+2	h1	9.33397E-4
e2	-1.46066E+5	h2	-2.56877E-1

Surge Limit Line		Surge Control Line	
s0	-2.80000E+1	c0	-2.80000E+1
s1	7.00000E+0	c1	6.66667E+0

Wegmann Coefficients

The Wegmann polynomial represents another attempt to characterize wheel map information using a simple polynomial.

$$Efficiency = w_0 + w_1\left(\frac{Head \cdot 10^5}{Flow^2}\right) + w_2\left(\frac{Head \cdot 10^5}{Flow^2}\right)^2 + w_3\left(\frac{Head \cdot 10^5}{Flow^2}\right)^3$$

Where

(w_0, w_1, w_2, w_3) are the Wegmann Coefficients

Rather than fitting Wegmann coefficients to the measured data, we fit them to the Cheesman coefficients in the region where the wheel map is actually drawn. This somewhat reduces the tendency of Wegmann's relationship to increase without bound as the flow decreases (we also limit the maximum efficiency to 100%).

This polynomial is used to calculate gas compression power lines using the relationship:

$$kWgas = \frac{Qmass \cdot Head}{Eff}$$

Using Wegmann's equation to solve for efficiency, and solving for mass flow using:

$$Qmass = \frac{Q \cdot MW \cdot P_b}{Z \cdot R \cdot T_b}$$

we can solve for head at any given flow and gas compression power.

To do this, Andromeda devised a fast-converging iterative solution, based on a linear predictor.

The input power requirement can be expressed in either kW or hp, depending on your current unit of measure preference.

Adiabatic Eff.W		BWR Eff.W	
w0	4.97981E-1	w0	4.97981E-1
w1	2.56849E-5	w1	2.56849E-5
w2	-5.97541E-10	w2	-5.97541E-10
w3	4.07460E-15	w3	4.07460E-15
wn	1.90600E+0	wn	1.90600E+0

Important Note

You can have any number of speed, efficiency, and power lines:

Wheel Map Range

Speeds	Efficiency %	Power
3200	80.00	5000
3500	83.00	12500
4000	83.52	20000
4500	83.00	
4950	80.00	
5200	75.00	
	70.00	
	65.00	

The speed and power lines can be entered in any order. Efficiency lines are entered from left to right, with the peak efficiency line being somewhere in the middle.

If you specify an efficiency greater than the maximum efficiency, the actual peak efficiency will be used. To find the maximum efficiency, click the "Redraw" button. The maximum efficiency for the wheel map (and the overlay wheel map, if you're plotting one) are calculated and displayed under the wheel map area:

Emax	0.000
Emax (ov)	0.000

Zero entries are ignored.

You need to specify whether to use the adiabatic or isentropic (BWR) coefficients to draw the wheel map:

Adiabatic	Main
Adiabatic	Overlay

You can also tell GasPerf whether to plot an overlay wheel map, and if so, whether to label each of its lines:

Labeling the Lines

When you redraw a wheel map, you have three choices for label placement, determined by the menu just above the "Redraw Map" button:

Redraw Map Prefs

Draw labels at saved positions

| Redraw Map | Done |
| Redraw Labels | Help |
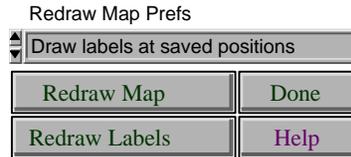
- Redraw labels at default positions

  The labels will be placed as follows: to the right of speed lines, above efficiency lines, and near the middle of power lines. The surge limit line is labeled to the left, near the bottom third of the line, and the surge control line is labeled to the right, near the upper third.

  In many cases, this is all you will need to do.

- Redraw labels at saved positions

  If you've previously saved this wheel map, you can redraw the labels at the positions they were in when you saved the wheel map. The graph range is also restored. If the saved position is too far from the default position, the default position will be used.

- Don't move labels on redraw

  Redraw the lines, but don't move any labels. This is handy if you want to redraw the lines with more line segments (to obtain a smoother line when you print the wheel map).

Set the view selector to "Move Labels" to change the label positions. You can either move the labels manually, or you can use the cursor controls.

Moving Labels Around

Start by making sure that the crosshair control is selected in the graph palette:

While you are moving labels, the control point for each label is visible as a small circle:

4500 RPM

○

You can move labels by simply dragging the control points with the mouse. Or, you can select the labels you want to move by selecting them in the cursor list. In the example below, we'll move the 4000 and 4500 RPM labels together (note that they are both selected):

3200 RPM	12.09	5.43	□	○
3500 RPM	13.10	6.91	□	○
4000 RPM	14.72	9.46	■	○
4500 RPM	16.69	11.61	■	○
4950 RPM	18.03	15.56	□	○
5200 RPM	18.90	17.53	□	○
80.0 %	9.88	37.95	□	○

Now you can move the selected labels by clicking the cursor arrows:

Colours and line styles

You can assign colours to the speed, efficiency, power, surge, and surge control lines by clicking a colour selector, and holding down the mouse button. A colour pop-up menu appears — select any colour you want from the list:
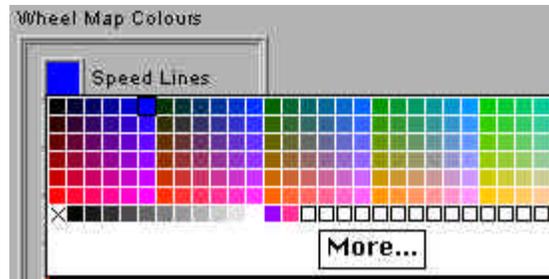
The changes you make don't take effect until you click "Redraw Labels", or switch views.

Overlay Wheel Map

In addition to the control available for the main wheel map, you can select the line styles to be used for each of the lines on the overlay wheel map:

Remember that overlay wheel maps are only plotted if you request them:

The Gas Composition fields in this area are used to compute the gas compression power lines:

Wheel Information

Wheel #	W127-36-1	MW	17.000
Manuf	C.E.S.	Pb	101.33
Booster #	SN 1650 PC	Pf	5200.00
Impeller #	RFB836	Tf	28.00
Curve #	6%	Z	0.9026
Diameter	953 mm	Test	1993-07-13
Station	Vetchland 2	Print	1996-10-20
			Test Notes

This area can be used for test notes, or for comments regarding any limitations in the wheel map (for example, that measurements were only made at a limited number of speeds).

Pb (base pressure) and Pf (flowing pressure) must be in kPa. Tf (flowing temperature) must be in deg C.

This is probably the view you would use for printed wheel maps.

Note
If the power lines are flat, it's typically because you haven't entered the gas composition data into this screen.

General X-Y Graph

The General X-Y graph lets you plot any measured or calculated signal against any other signal. You can place as many *plots* as you wish on an X-Y graph.

For each plot, you can display a fitted curve, the underlying data points, or both. The fitted curve can be displayed over an arbitrary X-axis range, including extrapolated ranges:
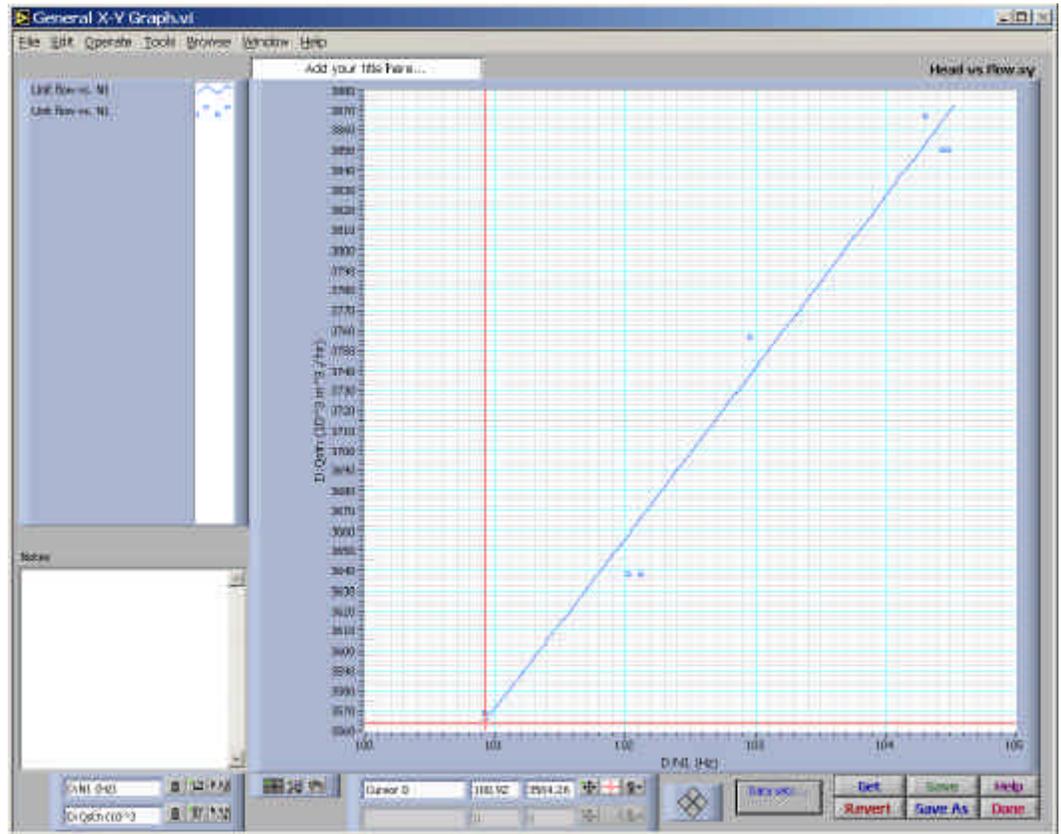
You can display the plots in a wide range of colours, line styles, line widths, and point styles. These can be changed by clicking on an individual plot in the *legend:*
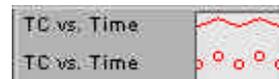
When you click and hold the mouse button down while the cursor is over the line or plot area to the right of a plot name, a pop-up menus appears, which lets you change the plot attributes (note that there are separate entries for the fitted curve, and for the underlying data points for each plot):
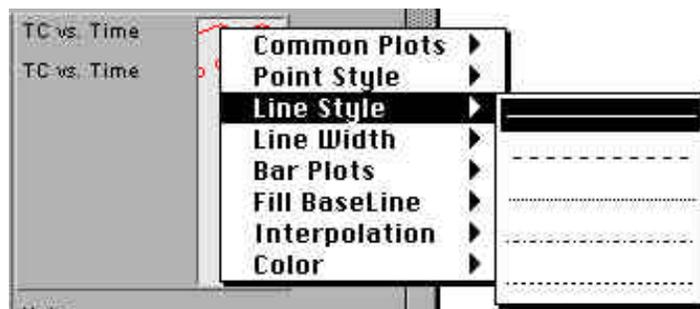
The buttons in the lower-right hand corner of the window have the usual meanings (remember that you can always show the floating help window to obtain help for a specific control):

To import data from the current *Snapshot Buffer,* or from a saved X-Y graph, click the *Data Set Details* button:

The *X-Y Data Points* vi which appears also lets you fit polynomial curves to the data sets, specify the range over which to plot the fitted curve, and specify whether to display data points on the X-Y plot.

X-Y Data Points

The *X-Y Data Points* vi is used to build X-Y graphs. It lets you import data from the current Snapshot Buffer, or from a saved X-Y graph. You can also fit polynomial curves to data sets, specify the range over which to plot the fitted curves, and specify whether to display data points on the X-Y plot.

This vi displays an array of *plots.* Each element of the array corresponds to an individual plot:
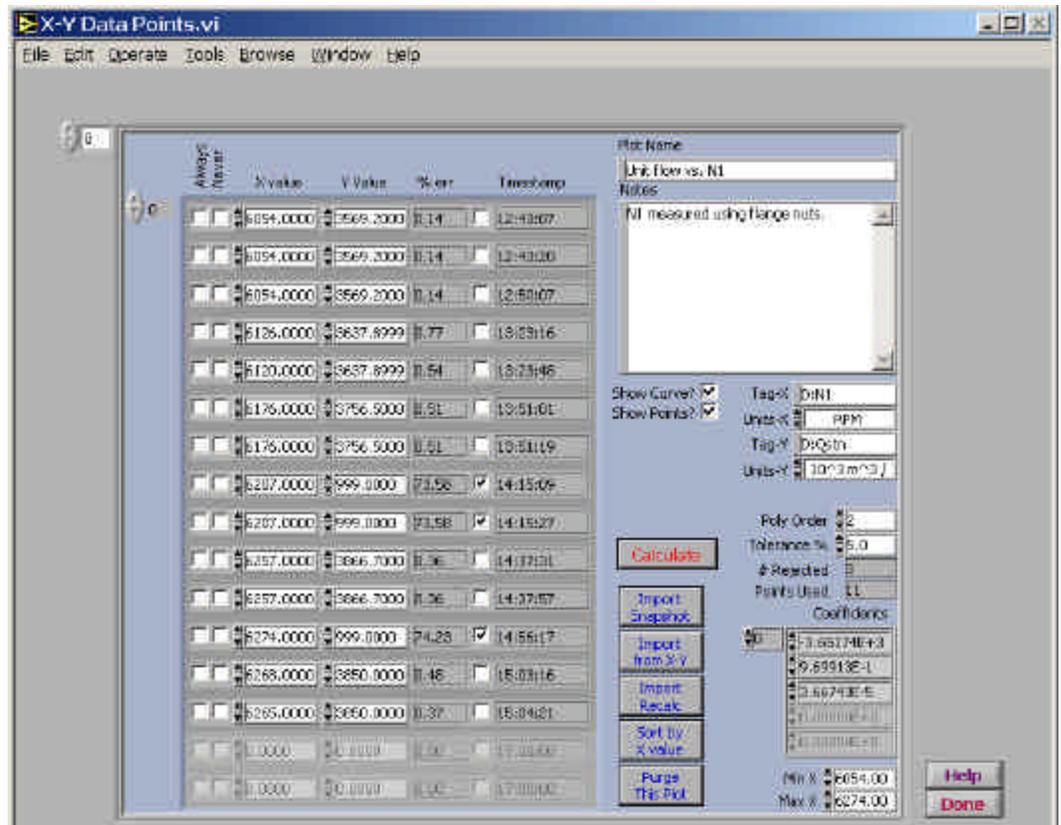
Like the configuration arrays, the plots displays are initially grayed out, indicating that there is nothing at the current position in the array. Click one of the control button for the plot to create a plot at the current index (zero in the example above). You can have any number of plots in an X-Y graph.

You can import a set of plot data from the current Snapshot Buffer:
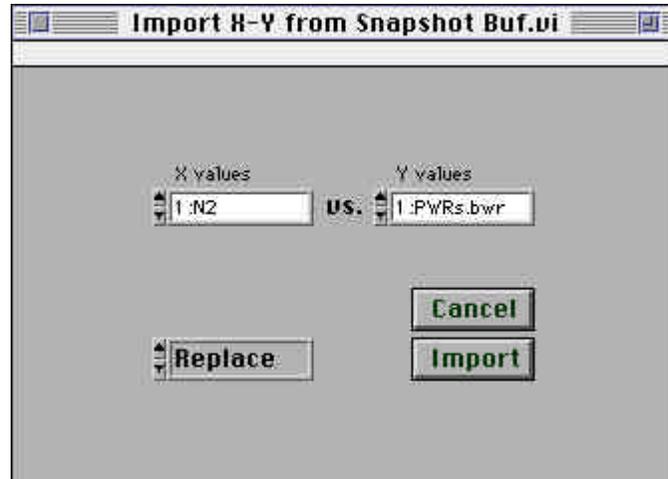
You can also import a plot from an existing X-Y graph. You are asked to locate an X-Y graph, and the following dialog appears:
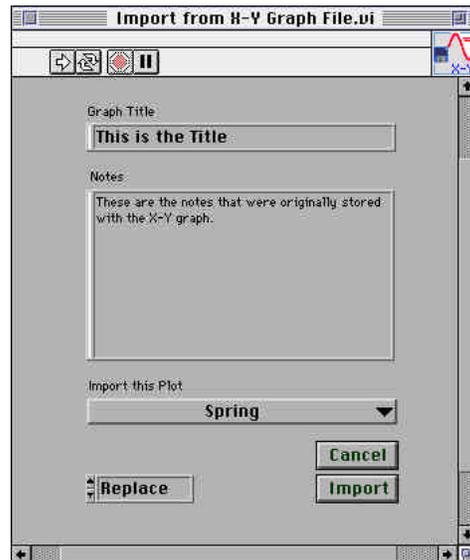
You can select the plot to import from the pop-up menu.

Additionally, you can import a data set from a recalc file:
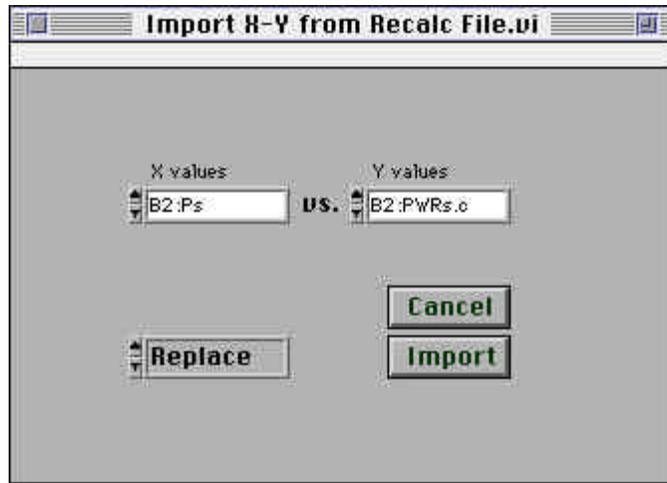
In each case, you can either replace the plot you see on the screen, or simply append new data points to it. When you *replace* a plot, attributes such as the X & Y tags and units are also imported.

The name you give to each plot (top right-hand corner) will be used to label the plot in the X-Y Graph legend.

You can fit a polynomial curve to the data points by clicking the "Calculate" button. You can specify a polynomial order from 0 to 6, and a tolerance percentage.

The curve fitting procedure is iterative: a polynomial of the specified order is calculated using a least-squares fit, and the SVD coefficient fitting algorithm. Following this, points which are more than Tolerance % from the fitted curve are removed, and the procedure is repeated until no more points are removed from the calculation. You can see how many points were used, and how many were rejected:
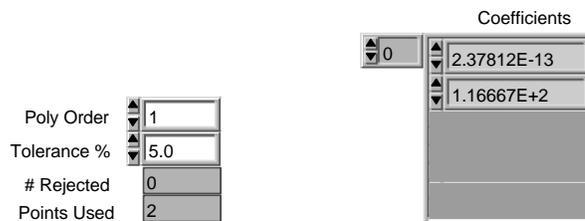
You can also control the point selection manually. Points which you specify as "Always" being included are always used in the curve fit, no matter how badly they fit the polynomial curve. You can also specify that points should "Never" be included in the curve fit.

For each point, you can see how far it is away from the calculated curve, and whether it was selected for automatic removal:
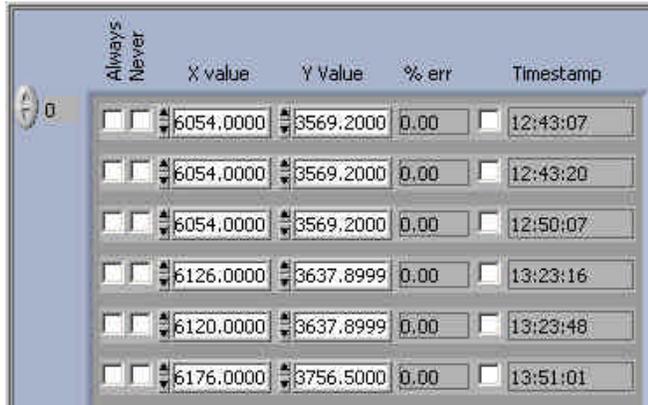
You can also manually enter new data points.

Extrapolation

This VI arbitrarily limits you to sixth-order polynomial curves, on the grounds that higher order curves will virtually always contain unwanted inflection points.

This becomes most evident if you extrapolate a polynomial curve beyond the measured data range — the behavior of the polynomial may have little or no relationship to the physical system.

It's generally a good idea to use a low-order polynomial for curve fitting unless you know that there is a physical relationship which requires a higher-order polynomial to model.

You can fit a polynomial to the data set for each plot. You can choose a polynomial order between 1 and 6, and a tolerance beyond which data points will be rejected (polynomial fitting works just the way it does for the wheel map).

Displaying Plots

For each plot, you can display the fitted curve, the data points, or both:

Show Curve? ■
Show Points? ■

For the fitted curve, you can select the range over which you wish to plot the polynomial:

Min X 0.00
Max X 36.00

When you calculate new coefficients, the Min X and Max X values are set to match the data set range.

Deleting Plots

You can delete the current plot (the one you can see) by clicking:

Purge
This Plot

Sorting

You can sort the current data points in ascending order of X-Value (this doesn't affect how the plot is displayed, but may help you to see relationships in the data points, or to find specific data points. Data points are initially imported in time stamp order):

Sort by
X value

X-Y Graph at Multi-Base Conditions

You can create an X-Y graph in which each plot shows the same X and Y signals, but calculated at different base conditions.

An example of this might be to plot power as a function of some other variable at different base temperature conditions:

You can create this X-Y graph using any X and Y signals in the Snapshot Buffer, but it doesn't make sense to plot variables which do not change with the base conditions (because each of the plots would be identical).

When you click "OK", the system starts to recalculate the Snapshot Buffer at each of the specified base conditions (a progress monitor tells you which one is being calculated).

When the calculations are complete, you see the "X-Y Data Points" vi. Here you can make any changes you want to the X-Y graph.

The software will initially try to fit second-order polynomials to each of the plots, with a 5% error tolerance.

When you're satisfied with the plot data, click "Done" to display the X-Y graph.

The X-Y graph produced by this VI is just like any other X-Y graph: you can save the graph, add data series, hide plots, and extrapolate the fitted curves as desired.

ASCV Capacity Graph

You can create an X-Y graph with a plot for each ASCV position in the Snapshot Buffer:

Data Sources
(Tag Names)

M9	Unit ID
HEADbwr	Head.
FLOW	Flow.
%ascv	%ascv.

Data Sources
(Pop-up Menus)

M9:HEADbwr	Head
M9:FLOW	Flow
%ascv	%ascv
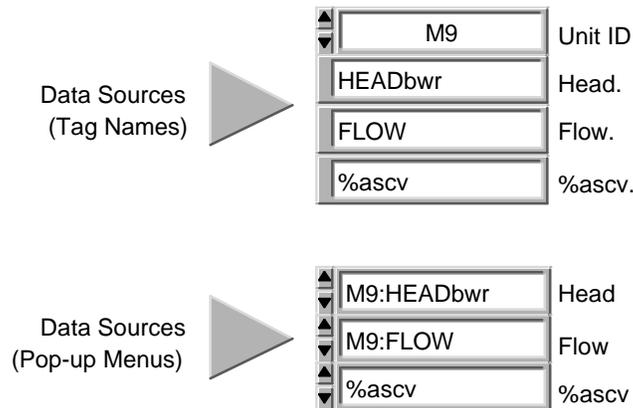
This VI will initially propose the BWR head vs. actual flow for the first unit, but you can use any combination of variables in the Snapshot Buffer. The value you use for "Head" will be the Y axis, and the "Flow" variable will be the X axis.

You can select the variables to be used in two ways:

▪ Type the desired tag name into one of the upper string controls (if the unit is the same as the UnitID shown at the top, they do not need to be qualified by a unit name).

▪ Select tags from the pop-up menus in the lower area.

The Snapshot Buffer will be divided into as many segments as there are ascv positions, and sorted by ascv position[8].

When you click "OK", the system will add the point (0,0) to each plot, and will attempt to fit a second order polynomial to each ascv position. The "X-Y Data Points" vi will be displayed: at this point, you can edit points, add plot segments, and calculate new polynomials to describe each plot.

Click "Done" to view the X-Y graph.

---

[8] This method assumes that the ASCV position will be manually entered (there presently is no provision to group values which are similar but not identical).

Importing Data Points

If you are not connected to the data acquisition hardware, you can import data points from a spreadsheet, by selecting "Import Pts" from the main menu. You are asked to located a tab-delimited spreadsheet file:
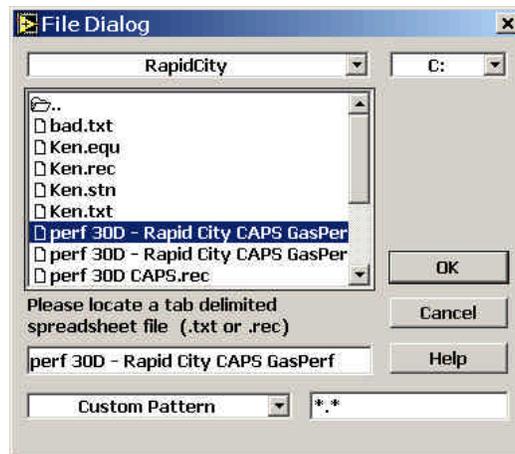
Your spreadsheet is displayed:
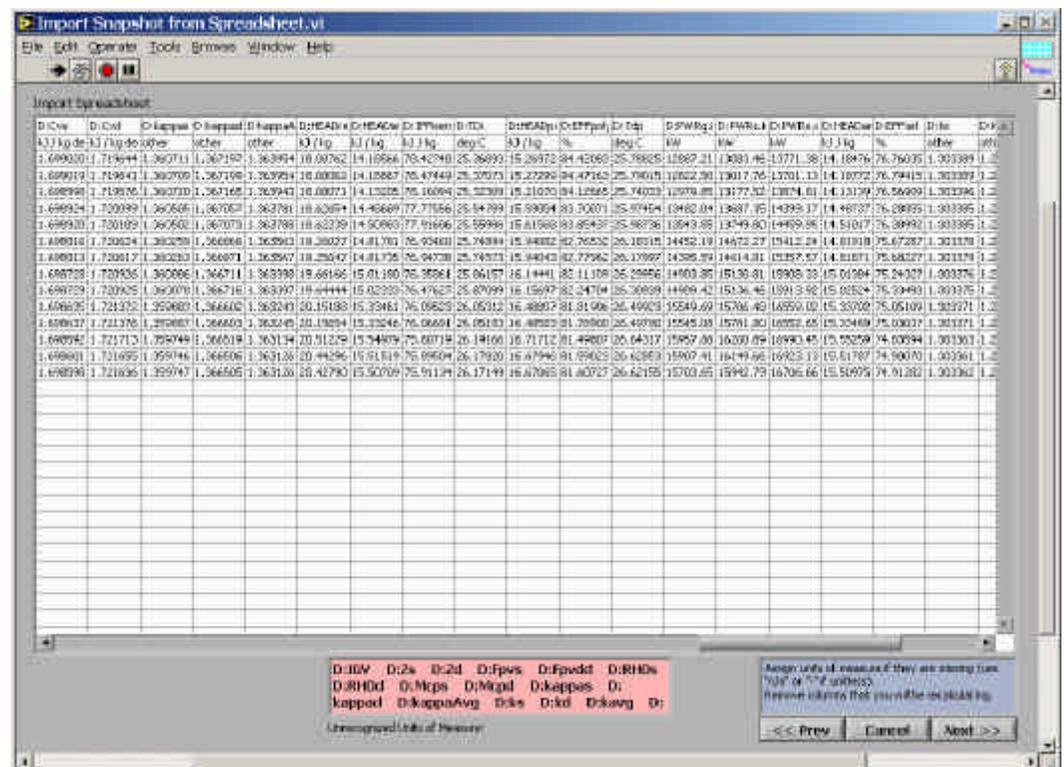
In the bottom right-hand corner of the window, you see instructions for each portion of the data import:

Assign units of measure if they are missing (use "n/a" or "-" if unitless).
Remove columns that you will be recalculating.

[<< Prev] [Cancel] [Next >>]

So there are two things we need to do before clicking Next:

1. Remove columns you don't need

   You might not need some of the columns in your spreadsheet, or they might contain values that you'd like to recalculate. To get rid of them, move the cursor to the upper border of the table. The cursor changes to a downward pointing arrow. Drag through the columns you don't need. The column grid turns blue, indicating that the columns are selected. Then right-click, and select "Cut Data" from the pop-up menus which appears.

2. Assign units of measure

   The second row is reserved for units of measure. You can assign any SI or U.S. Customary unit of measure which GasPerf can display. The list of unrecognized units of measure is kept up to date:

   D:IGV   D:Zs   D:Zd   D:Fpvs   D:Fpvdd   D:RHOs
   D:RHOd   D:Mcps   D:Mcpd   D:kappas   D:
   kappad   D:kappaAvg   D:ks   D:kd   D:kavg   D:

   Unrecognized Units of Measure:

   Some column, like Date and Time, won't have units of measure.

When you're removed the columns you don't need, and everything has a unit of measure that needs one, click Next:
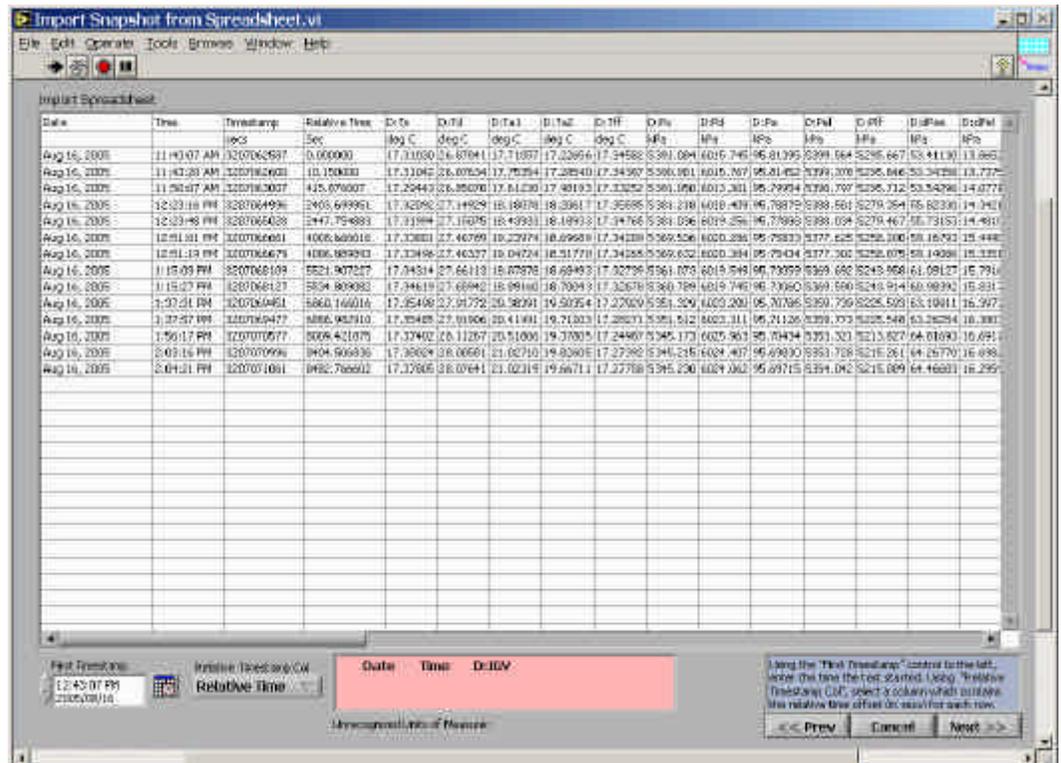
At this point, you need to tell the system what the timestamp for each row of data should be. Make sure that the First Timestamp, at the lower left-hand corner of the window, contains the time for the first row of data. If your spreadsheet was created by GasPerf, it will be correct already. If not, change it to the right value (you can click the little calendar icon beside the field if you want an easier way to enter the date).

You also need to identify a column which contains the relative timestamp for each row of data. Again, this will probably be set correctly if the spreadsheet was originally created by GasPerf. If not, select the column you want to use from the pop-up menu:

When you click Next, the spreadsheet is converted to SI units of measure, and displayed:
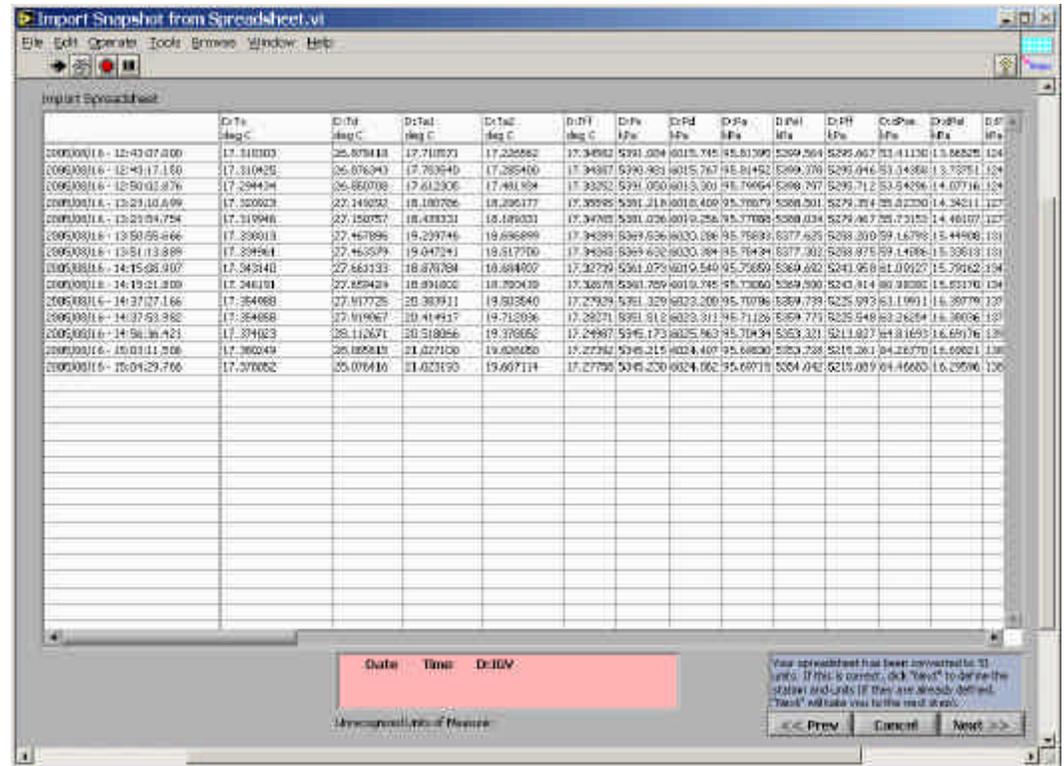
Click Next if you're happy with the conversion.

You now see a mini-menu, similar to the main menu screen:
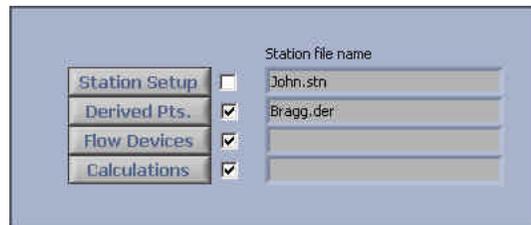
You MUST define a station (you can also do this before entering this wizard). Everything else is optional (you may get some warning messages if you don't define everything, but you can ignore them).

Note that the tags for all calculated results must be unique – if you recalculate something that is already in the spreadsheet, you will get "duplicate tag" error messages.

Click "Next" when you've define the calculations you wish to perform.

The spreadsheet is calculated, and you can optionally save it.

The Snapshot Buffer is populated with the contents of the calculated spreadsheet, so you can use all of GasPerf's strip charts, operating graphs, etc. to view the results.

Recalculate Snapshot Buffer

The recalculation function provides a powerful way to correct measurement errors, change gas conditions, experiment with different base or ambient conditions, or export data to another system.

From the initial screen you can change the scaling of any signal acquired by the PXI chassis or manual input. You can also change the gas composition, or reconfigure the flow devices:

> Gas Composition
>
> Flow Devices

You can also specify new base conditions. This affects all of the flow calculations, the equations of state, and the ascv capacity determination:

New Base Conditions

Pb (kPa)	Tb (C)
101.325	15.00
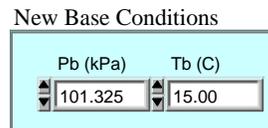
When you click "Recalc", you will be given an opportunity to change any of the manually input data points before the calculations are performed.

Recalculate Scaling
However, to correct any calibration errors, the system needs to know both the original scaling and what it should have been. The initial recalculation screen shows the original scaling for all PXI and manually entered signals, and proposes the same scaling in "Revised Scaling":

Original Scaling

Tag
M9:A0

Min raw	Max raw	
0.00	5.00	(raw units)

Min Eng.	Max Eng.	Units
0.00	500.00	kPa

Revised Scaling

Min raw	Max raw	
0.00	5.00	(raw units)

Min Eng.	Max Eng.	Units
0.00	500.00	kPa
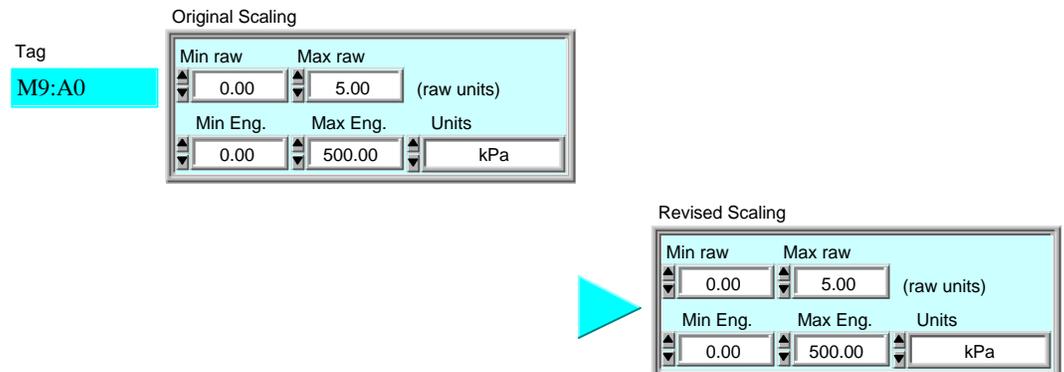
If you need to change the scaling for a signal, just modify the Revised Scaling here (don't change it in the PXI configuration screen)[9].

---

[9]   This is important — if you change the scaling in the PXI configuration screen, the scaling will not be changed, because these values are only used when new measurements are acquired.

Edit Manual Points

All of the data values acquired by the PXI chassis or by manual input can be modified:

Data Points   Unit+Tag

0						
0	0.00000	5.50000	10.00000	Relative Time
	14000.00000	13995.00000	13998.00000	M9:N3
	95.60000	96.00000	96.50000	M9:TC-1
	14.56000	15.01000	14.75000	M9:RTD
	0.00000	0.00000	25.00000	M9:%ascv
	15.00000	15.00000	15.00000	M9:Ta
	95.00000	95.00000	95.00000	M9:Pa

You can scroll by row by changing the upper array index (you can also click the little up/down arrows). The lower array index scrolls the columns.

The list of tags at the right-hand side of the screen will scroll along with the rows.

Edit Recalc Points

Finally, the system recomputes all of the calculated values (flows, powers, etc.), and displays the results in a table:

Recalc Buffer

Date	(n/a)	Apr 22, 1997	Apr 22, 1997
Time	(n/a)	10:17 a.m.	10:17 a.m.
Timestamp	Secs	147219856	147219861
Relative Time	Secs	0	5.5
M9:N3	RPM	14000	13995
M9:TC-1	deg C	95.6	96
M9:RTD	deg C	14.56	15.01
M9:%ascv	kPa	0	0
M9:Ta	kPa	15	15
M9:Pa	kPa	95	95

You can scroll in any direction by using the scroll bars (the display updates while you move the "thumb" of the scrollbar). The left-most column (tags) does not scroll (it acts like a frozen pane in a spreadsheet).

You can edit any cell in this table.

You can save this Recalc Buffer as a tab-delimited text file suitable for import into a spreadsheet or database.

If you want to re-import the recalc file back into GasPerf, we suggest that you avoid changing the units of measure.

Appendix 1 — Controls & Indicators

Each window in the Performance Testing software is sometimes referred to as a "virtual instrument" (VI), because it acts somewhat like a real instrument might act. You can use the mouse to push buttons and turn knobs, much as you might with a physical instrument. There are two main kinds of objects that can be on a screen: *controls,* and *indicators.* You can change a control; an indicator just displays information.

Here is how they work:

Digital Numeric

Min raw

0.00
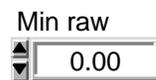
Double-click the mouse within the numeric area to highlight a new value (if you pressed the Tab key to navigate to a nume number is already highlighted).

Alternately, click and drag to select part of the number, and t the portion of the number. You can also click the little arrows the value.

You can also press the up and down keys on the keyboard to To change the value more rapidly, press the shift key while yo

In some cases, there is a limit to the range of numbers you can attempt to exceed the allowable range, the display will reset it allowable number.

In the Performance Testing system, a control has a white bacl have a gray or coloured background.

Buttons

VXI Setup

Derived Points

Flow Devices

Calculations

Config Report
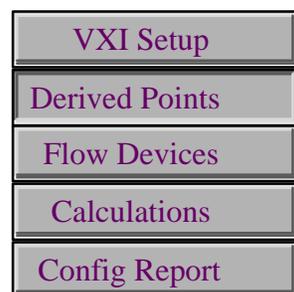
Buttons come singly, or in groups. Click to "depress" a buttor like the "Done" button which appears on every window, only until the program has noticed it.

Some button have keyboard shortcuts:

Help button F1 key

Done button   Enter key (on Help screens only)

Checkbox

☐ Fast Scan

Click inside a checkbox to turn it on or off. When an 'X' app[ears], selected.

Some checkboxes are just for your information. For example, a series of checkboxes to remind you of which configuration s[teps] already taken.

Slide Switch

On
Off

Either click the desired value, or click and drag the indicator t[o the] position. If the two positions aren't labeled, the top position i[s On].

Pop-up Menus

Units
Hz
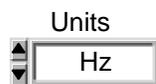
Pop-up menus let you make a choice from a list of possibilitie[s]. within the text area, and hold the mouse button down.

A menu of all of the choices pops up — move the cursor to y[our] release the mouse button.

Units
✓ Hz
Rev / sec
RPM
Secs

You can also click the little up/down arrows to change the sel[ection].
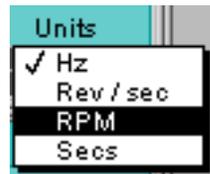
View
Frequency ▼
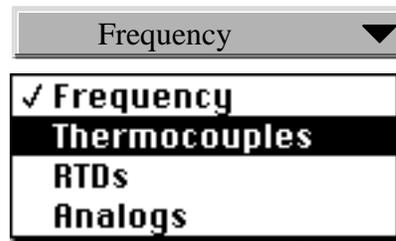
Pop-up menus can also look like this.

✓ Frequency
Thermocouples
RTDs
Analogs

Clusters

Several items can be grouped together in a Cluster.

In the example below, we see a cluster which contains three th Name), and two numbers (Pb and Tb).

Plot Name	Pb (kPa)	Tb (C)
Plot One	95.000	12.00

You can move from field to field by pressing the Tab key.

You can copy the contents of an entire cluster to the system c them into a similar cluster. To do so, right-click on the border pop-up menu appears, from which you can select "Copy Data

To paste the values into a similar cluster (or another element click the border of the destination cluster, and select "Paste D up menu.

Arrays

In many cases, configuration options are contained in arrays. contain zero or more items.

In the example below, each element of the array contains a clu have arrays of numbers, etc.).

There is currently one element in the array — the upper box. two more boxes which have been "grayed out". These are plac items. To create a new item, simply click inside one of the ele and enter something. The item will stop being grayed out, and becomes a new array element.

Base Conditions
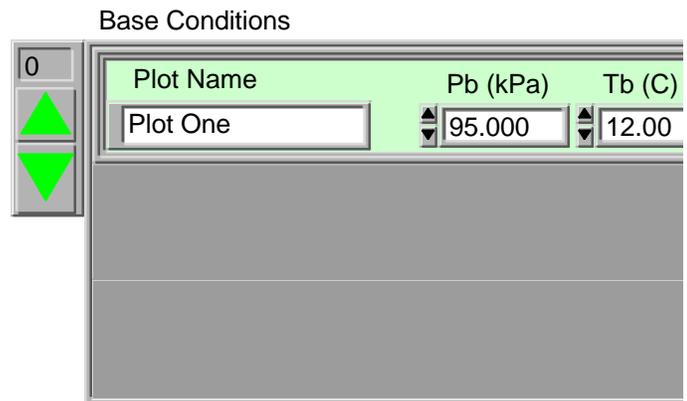
There may be more array elements than can fit on the screen. through the array elements by clicking on the big arrows in th corner of the array (the two arrows and the optional number index.

If you want, you can enter a number in the array index to mak immediately jump to that element. The first item in an array zero.

Graphs

The strip charts, wheel maps, and X-Y graphs all offer you a great deal of control over their appearance.

For starters, you can change the range of the X or Y axis by selecting the first or last number in the graph scale, and changing it:
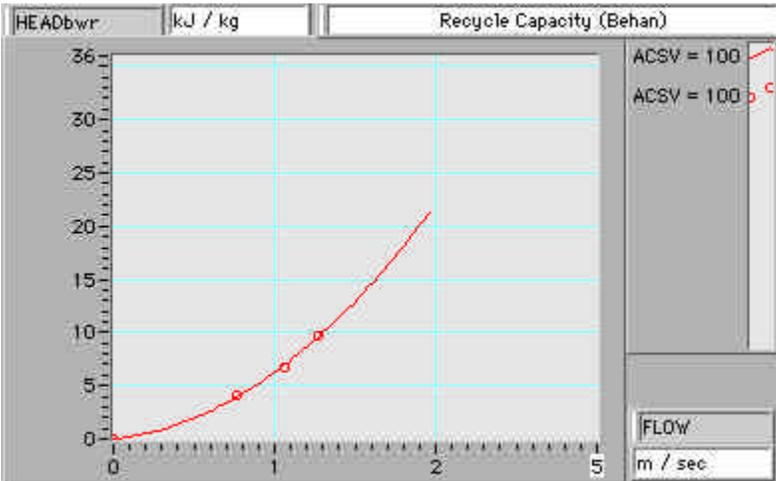
When you click elsewhere or press Enter, the scale is re-calculated with the new limits.

If you right-click on an axis scale display, you can even decide to place the markers at arbitrary positions:
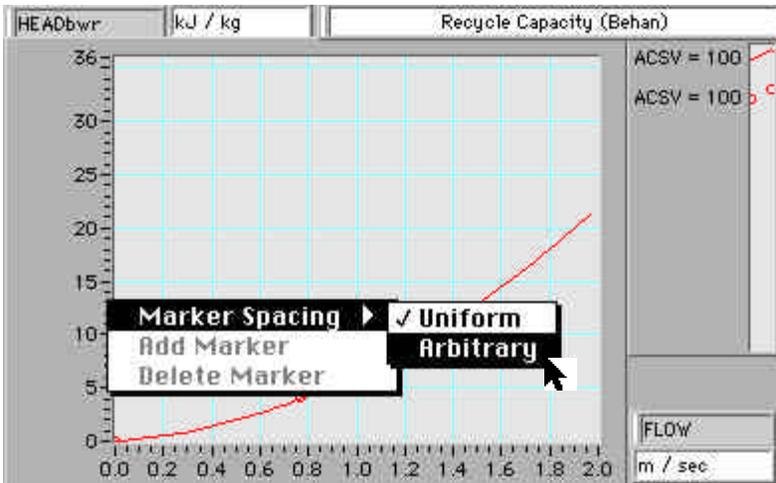
Now, you can make the markers any value you want, or else drag them to the exact location you prefer.

There are some more options available if you right-click with the graph body:
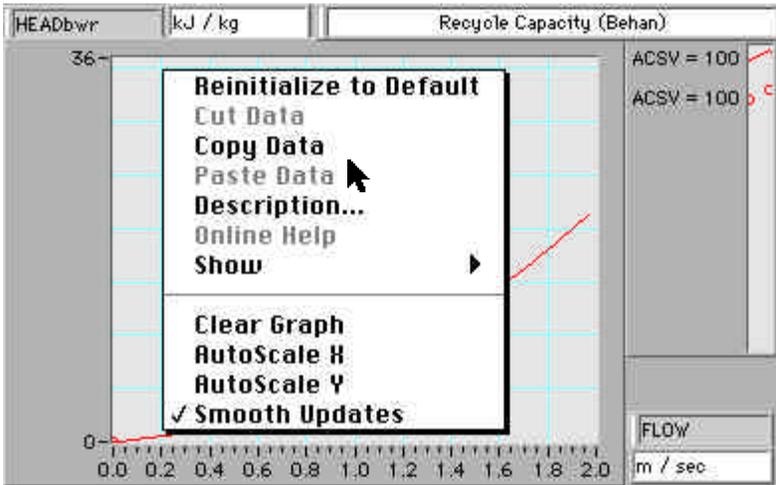
If you want to copy the graph to another application, you can select "Copy Data"[10]

The Palette Display

Most graph have a "palette" associated with them:

The two slide switches at the left turn autoscaling on and off (on is to the right).

The next column contains two pop-up menus, one for the x-axis and one for the y-axis, which let you control the graph scales. Perhaps the most common operation is to set the digits of precision to display:
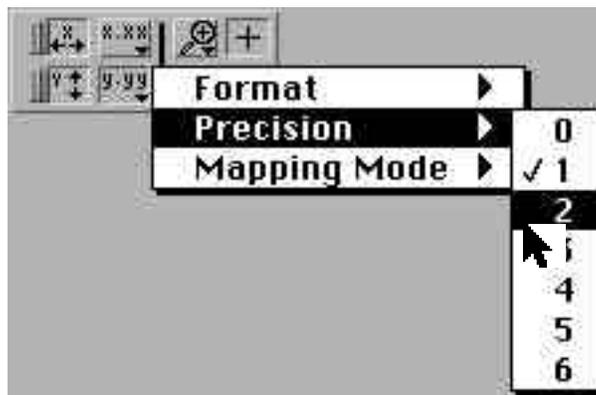
You can also make the scaling logarithmic, or display the scale in scientific notation.

---

[10]   On the Operating graphs, you should be aware that the wheel map and X-Y graphs are actually a "stacked" set of graphs, and that the top graph just contains the operating point history. This is the graph you'll get if you select "Copy Data". Also, most graphs display a graph legend and / or palette, which will also be copied unless you turn them off (use the "Show" slide-out menu).

Next to the right is the little magnifying lens . If you click this icon, the cursor turns into a descriptive icon, and the most recent zoom mode is selected. If you click and hold, a pop-up menu appears which lets you select a variety of zoom options:
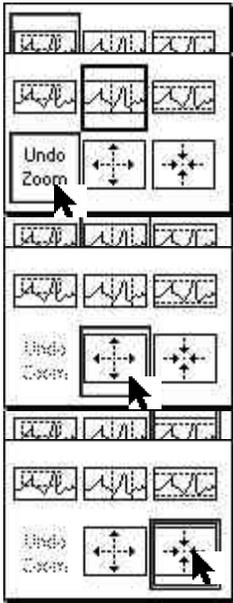
The "grabbing hand" icon lets you pan around the graph. You just "push" the graph to where you want it.

The crosshair icon is used to move cursors around on the graph.

The Cursor Display   Many of the graph also have a cursor display:

You can create a new cursor by clicking anywhere in the "greyed-out" area. To remove all cursors, right-click the border of the cursor display, and select "Empty Array" from the pop-up menu which appears.

You can name the cursor, and move it around by typing values into the X and Y axis displays. If you move the cursor by dragging it (remember that the crosshair must be selected in the palette), the X and Y axis displays show the current position of the cursor.

The next box determines whether the cursor will be moved by the arrow cluster:
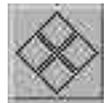
All of the selected cursors are moved together.

If you click and hold the mouse button on the next box over a pop-up menu lets you change the appearance of the cursor:
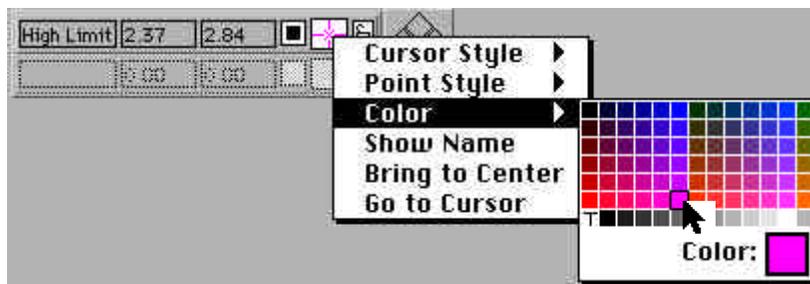
The "Bring to Center" option is particularly handy if you can't see the cursor.

Finally the lock icon is a pop-up menu which lets you lock the cursor onto a plot, or onto the data points which make up the plot: