GASWorkS™ 10.1
User's Manual
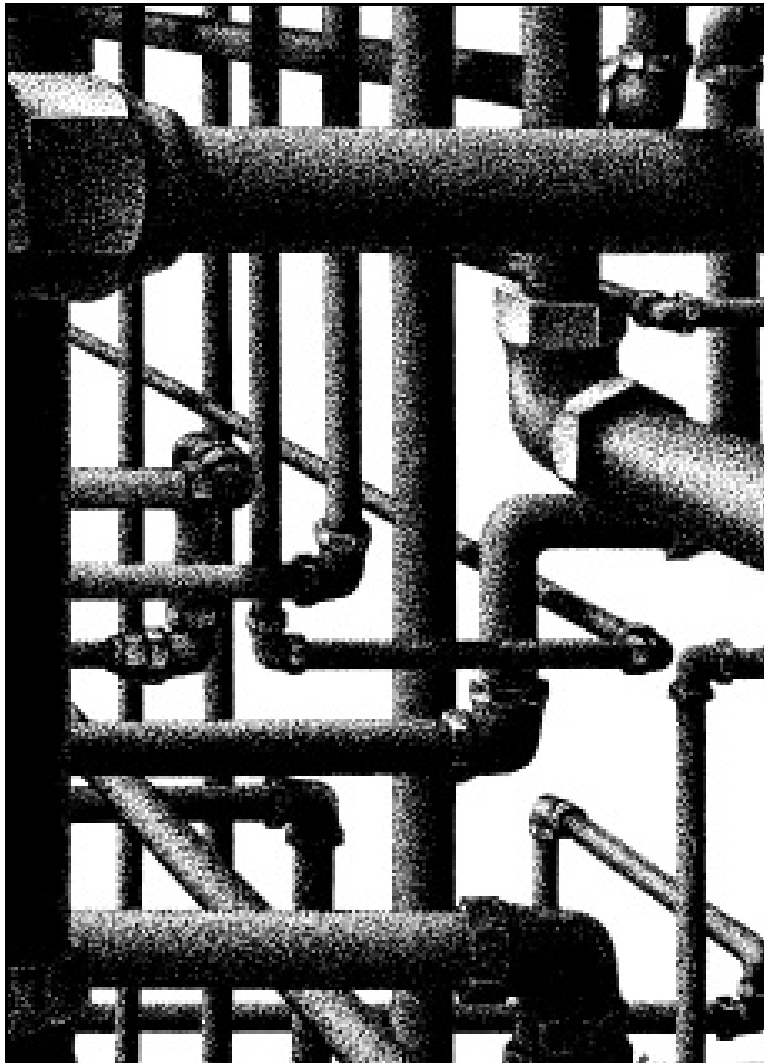
GASWorkS™ 10.1

License Agreement

GASWorkS™, its options, and supplements are licensed for installation on only one computer per authorized copy. That is, only one copy of the program is intended to be installed per license. Each purchased copy of the program represents one license. We rely on your honesty and integrity to ensure that multiple copies of the program are not created.

In reference to this paragraph, the following definitions shall apply: Software shall mean the GASWorkS program, documentation, and any associated options, enhancements, or supplements; User shall mean the individual or organization purchasing the Software, the individual or organization employing the persons to be using the Software, the individual or organization otherwise responsible for the administration of the use of the Software, or any affiliate thereof; Developer shall mean B3PE LLC. By use of the Software, the User consents to accept full responsibility for the use of the Software, including Determination of fitness for purpose; Interpretations or decisions based on any results derived from the Software; Any claims resulting from the use, accidental misuse, or intentional misuse of the Software by the User; And for reimbursement of the original purchase price to the Developer for each unauthorized copy of the Software generated from the User's authorized copy.

By installing the Software, the User consents to the means of copy protection.

It is the User's obligation to immediately return the Software, without installing it, if they do not agree, concur, and consent to these Licensing requirements.

Trademark Notice

GASWorkS, GASCalc, GASBase, and the "B-Cubed" logo are trademarks of B3PE LLC. dBase is a trademark of ASHTON TATE, a division of Borland International. AutoCAD is a trademark of Autodesk, Inc. Windows (and its variations and derivations) are trademarks of Microsoft Corporation. MicroStation is a trademark of Bentley System Inc.

Copyright Warning

The GASWorkS software and User's Manual are protected under United States and international copyright laws and treaties. Making unauthorized copies of the software or the associated documentation violates United States and international law. Don't be a criminal; don't copy.

GASWorkS™ 10.1

Guarantee

If you are not fully satisfied with the performance of GASWorkS, uninstall the software and return the original distribution materials, security device (if provided), and documentation within thirty (30) days of the date of receipt. We will promptly refund the original purchase price.

If during the useful lifetime of the software, GASWorkS or its options do not perform as claimed, notify us of the deficiency, and we will promptly address your concern.

About The User's Manual

This User's Manual attempts to provide the User with an understanding of the GASWorkS software by providing extensive descriptions and documentation of its various features. The main body of the User's Manual presents descriptions of the various menu items and data screens. Separate sections are provided for several of the more extensive features. The *Appendix* section presents a primer on network analysis, along with technical documentation of the various equations and data files used and supported by GASWorkS.

In addition to the information contained in this manual, examples, how-to tips, and instructions are included in the GASWorkS Help Guide.

A copy of the User's Manual in Portable Document Format (pdf) is contained in the GASWorkS installation file. With some versions of GASWorkS, the Demonstration Guide, User's Manual, and Help Guide are installed and viewable using the *Help* menu. Additional information and documentation may be found on our website, www.b3pellc.com.

GASWorkS™ 10.1   *Table Of Contents*

GASWorkS™ 10.1   *Table Of Contents*

GASWorkS™ 10.1 — *Table Of Contents*

GENERAL
INFORMATION

GASWorkS™ 10.1   *General Information*

Introduction

Use the procedures outlined in this section to install GASWorkS and begin using it to model your gas system. If you are a new User, we suggest working on the example model in the Demonstration Guide. The Guide will provide instructions and examples of using the basic features of the GASWorkS software. If you are a current User, you might want to browse the *Release Notes* topic in this section to review the new features contained in this version of the software.

A list of the available commands in the Graphic Data Interface (GDI) is provided in that section of this manual. The *Menu Items* section provides some abbreviated instructions on accessing and using many of the routines found in the software.

GASWorkS is suitable for modeling various distribution, gathering, transmission, and plant piping systems containing almost any compressible, single-phase fluid. Many of the examples used in its documentation refer to distribution-type systems. However, please consider that the methods required to create and solve these examples are the same as those required to create and solve a model of any type of system.

If you encounter problems or have specific questions, refer to the appropriate section of this Manual for detailed instructions, descriptions, and explanations. Or, use the Help Guide to view examples and how-to tips on performing a specific task.

Feel free to contact us with your questions. Additional support, documents, new and updated Property Tables, program updates, and frequently asked questions and answers can also be found on our website, www.b3pellc.com.

System/Hardware Requirements

GASWorkS will operate on any device configured to run Windows 8 through Windows 11. Best performance will be achieved using a device with elevated memory size and processor speed. The use of a mouse or similar pointing device is recommended. The use of a touchpad or similar point devices is discouraged.

The GASWorkS software will require a minimum of 37 MB of free hard disk space when installed. An internet connection is required for the license management and the use of the GeoMap functions.

B3PE LLC   *Engineering & Software Services*

GASWorkS™ 10.1　　　　　*General Information*

Licensing System

GASWorkS uses an internet-based licensing system. The software can be installed on any number of devices. License information is stored on our server. When a User attempts to use the software, the license availability is checked on the server. If the number of "in-use" licenses is less than the purchased number of licenses (the User Limit), the User is allowed to continue; otherwise, the use of the software is denied until the number of "in-use" licenses is below the User Limit.

A license can be "checked out" for up to thirty (30) days. Once "checked out," the license is unavailable for use by any other Users until the license is checked back "in." An internet connection is not required to use the software during the checkout period for the device and User that checked out the license.

GASWorkS Data Elements

GASWorkS uses an intuitive set of data elements to represent the piping systems that it models. The basic elements of its data structure include node, pipe, and customer data. Nodes represent the connections, intersections, and terminations of pipes. Nodes are the points where flow is introduced into the system (supplies) or taken out of the system (demands). Nodes can also be used to represent customer connections and to assign customer loads. Customer features provide an efficient and intuitive way of representing customer locations and demands.

Pipes connect the nodes and carry flow between them. Flow may be thought of as going into or out of nodes. However, flow only goes through pipes. GASWorkS supports various hydraulic features -compressors, fittings, line pipe, regulators, valves, and wells.
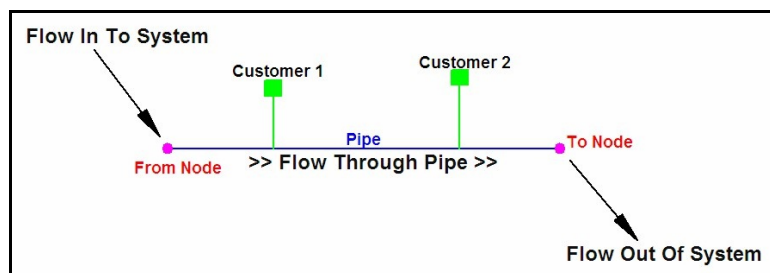
Both hydraulic data and attribute pipe data may be maintained. The hydraulic pipe data represents the items that affect the flow and pressure drop through the pipes. For example, pipe diameter, length, connectivity, etc. The attribute data represents items that do not affect the flow or pressure drop through the pipes.

The customer data contains information about the location and demand of the customers associated with a model. The customer data can be divided into two types of information - model data, which describes the customer's location, the supply main identity, the node load application details, and the total customer load - and a second type of information called attribute information, which can contain a variety of non-model data associated with the customer.

GASWorkS™ 10.1   *General Information*

All of the model files are accessed by way of a single "key" file known as the "Header" file (.hdr). This file contains general information about the model and the solution control parameters and serves as the access point for all other files.

Various other data are also used to support GASWorkS and its various routines. These include gas properties, User Text and Graphics, and background images. Each is described in detail in the various sections of this manual.

Notes On Dimensional Units

When the standard version of GASWorkS first starts, its default dimensional units are set to customary US values (Feet, Inches, Psi, Fahrenheit).

To change the default dimensional units, select the *Set Defaults* item from the *Utilities* menu. The Default Data Values screen is displayed. Select the *Dimensional Units* data tab to access the currently selected dimensional units. Make changes by selecting the desired item from the associated lists. Click the *Apply Dimensional Units Only* command button to change the values used in the current model. *Note* - Do not click the *Apply All Values* command button unless you are certain that you want to apply all of the default setting values on the various data tabs to the current model.

As used in GASWorkS all pressure units represent "gauge" units unless otherwise specified. Within the documentation, gauge pressure units are generally shown without a suffix; absolute pressure units are shown with an "a" or " (Abs)" suffix. Gas volume values or volumetric flow rates are expressed in "standard" cubic units, based on (adjusted to) the specified base pressure and temperature.

Notes For International Users

GASWorkS attempts to accommodate the needs of International Users by providing a robust set of dimensional units. These units can be selected as appropriate for the User's particular needs.

GASCalc does have a particular weakness when it comes to the various Property Tables that contain predefined compressor, fitting, pipe, regulator, relief, valve, and well data. These lists are primarily oriented towards US manufacturers and sizes. The Property Tables can be easily modified by the User to more specifically meet the particular needs of each User. The Property Tables provided with the software are intended to provide a starting point to allow the User to create their own custom Property Tables.

Note... | The example models provided with GASWorkS generally use US units of measure. To obtain valid results, these examples must be worked using the US units. When working these examples, if the default data values and dimensional units have been changed, they must be reset to their original US values before working with the example model.

GASWorkS can accommodate either a comma (,) or a dot (.) as the decimal identifier or a comma (,) or a dot (.) as the digit grouping identifier, depending on the associated Windows setting. For example, ten thousand and one tenth may be displayed as "10,000.10" or "10.000,10" depending on the Windows setting.

Use the Regional and Language options in the Windows Control Panel to change the digit grouping and decimal identification settings.

Release Notes

GASWorkS 10.1 represents a continued evolution of the GASWorkS software - however, as with all evolutionary processes, things change. Additional GDI Commands and functions were added, and inevitably, some features were discontinued. A partial summary of the many feature changes is provided in the following tables.

New & Enhanced Features
Some of the new and enhanced features available in the current GASWorkS release are listed (in no particular order) in the following...
A more efficient Graphic Data Interface.
Support for zoom and pan with the mouse wheel.
Click to edit or view data.
Use of "grips" to move features and data text.
Improved data viewing and editing.
Improved screen layout.
Improved access to the GDI Command List and Tool Palette.
Support for "User" Graphic Images - display icons, pictures, or pdf documents in the model.
Enhanced data text handling - easily move text and automatically include leader lines.
Display User-defined tables in the model.

GASWorkS™ 10.1   *General Information*

New & Enhanced Features
Display pre-defined User Text in the model - include standard notes, etc.
Supports "associated" User Text - link a User Text feature to a node, pipe, or customer data value.
Add User Text and include a leader line with a single command.
Supports import and export of KML format files - share data with Google Earth or compatible applications.
Performs coordinate projections to and from world coordinates based on standard ".prj" or ".wkt" files.
Includes support for ".jpg," ".pdf," ".tif," and ".kml" format files as background images.
Support for 3D coordinates and Isometric display.
Convert a line entity in a vector-based background image to a pipe in the model - makes updating a snap.
Supports multi-line User Text.
Support for node "attribute" data - especially useful for updating node loads.
Improved isolation valve handling.
Enhanced tracing - added additional options and color mixing for overlapping trace routes.
Improved multi-edit handling - easily update data or execute a command for a selection set.
Enhanced Facility Type handling - assign pipe flow equation or pressure units by Facility Type.
Enhanced default data handling - use specific default data or use the data from the last data feature.
Quick Solve routine - single-click solution makes solving and viewing results more efficient
"Favorite" results - allows quick review of results for User-specified features.
Speed enhancements for nearly all commands and routines.
Improved data list handling for User-defined databases.
"Csv" export - create a selection set and create a spreadsheet-compatible file with a few clicks.
Quick Solve from a report - solve from a report screen and view updated results without reopening the report.
Enhanced System Summary Report - more reporting options.
Date and time stamp for model data features so you can tell when an item was last changed.
Edit log to track changes to the model.

New & Enhanced Features
"Item" Summary Report to find and report item counts and quantities based on a User-specified selection set.
Cut and paste property data (pipe, fitting, etc) to/from the Windows clipboard.
Lock a Property Table to prevent unintentional data changes.
"A to Z" sorting of any item in a Property Table.
Cost and Part Number fields are included in all Property Tables.
Support for "power" type pipe flow equations - allows the use of CUST piping.
Support for "Fixed Pressure Drop" and fittings with pressure loss based on a pressure drop equation.
Enhanced regulator handling - calculates droop, handles low inlet pressure, and backlog switching.
Enhanced compressor handling - handles high inlet pressure and backlog switching.
Enhanced data text settings - separate settings for each feature type.
Model comparison routine - list the differences between the two models.
Copy User Text, User Graphics, and Saved Views from another model - share these items between models.
Connect loose pipe ends - select a group of unconnected pipes and automatically connect them.
Import customer service lines from "dxf" and "SHP" files.
Enhanced all import, merge, and export routines.
Copy and multi-edit User Text and User Graphics.
Copy a customer or pipe feature.
User Text, User Graphics, and pipe features may be grouped - think of it like a CAD block.
Added group handling commands - copy, move, delete, create, dissolve.
Added a "hydraulic status" to the node data - allows nodes to be turned 'on' or 'off' for tracing.
Added a "hydraulic type" to the node data - especially useful for tracing.
Use a "template" file when exporting "dxf" data.
Enhanced feature "snap" handling.
Supports display of a background grid and grid snap.

GASWorkS™ 10.1 *General Information*

New & Enhanced Features
"Quick Size" routine - set up a sizing specification and run it with a single click.
Added a *Snap Loose Pipe Ends* command to automatically connect pipe ends within a specified tolerance.
Added a *Straighten Polyline Pipe* command to remove vertices within a specified region.
Improved "unbroken intersection" handling - mark within a specified area, exclude nodes from review.
Improved symbol display handling - specify an on/off limit.
Support for multiple "undo" handling.
Redo handling.
Enhanced shapefile (SHP) import handling.
Enhanced DXF export handling.
Improved overall User Interface.
Added routine to create a load summary table, especially useful for documenting mechanical-style designs.
Enhanced Bill-Of-Material creation.
Added routine to automatically create a legend of User Graphic Symbols.
Symbolize fitting style hydraulic elements.

Discontinued Features
Some features have been discontinued from the current GASWorkS release. Their exclusion was based on their perceived usefulness concerning the latest software functionality. These features include...
Relate Features - The various relate features have been discontinued. These features were initially developed to help support the modeling of large multi-pressure tier systems. Modern machines offer enough power and memory to make these features unnecessary.
Digitizing Tablet Support - The various commands used to support data entry using a digitizing table have been discontinued.

We hope that you will be pleased with the many improvements and enhancements we have made to GASWorkS 10.1. Please let us know if you have any comments, concerns, or questions regarding our latest release.

Getting Help - Contact Us

Hours: Our normal work hours are 8:00 A.M. to 5:00 P.M. Mountain Time, Monday through Friday. However, if you have a problem, please try anytime - we may be in.

Telephone: (719) 578-9391

Email: help@b3pe.com

Website: www.b3pellc.com

MENU ITEMS

General Information

This section provides a general reference for using the GASWorkS menu bar and the associated menu items. Grouped by functional topic, procedures are described for using each of the menu items available in the GASWorkS menu bar. Most of the menu items will launch a data or specifications screen. Further information describing the use and features of the screens is provided in later sections of this Manual.

Certain menu items are not enabled unless a model is open.

File - *File Management*

The GASWorkS model file management routines are grouped in the *File* menu. The routines located in this menu are used to manage the model files only. That is, these routines manipulate the files directly associated with a model. The contents and use of each of the GASWorkS files are described in the *Appendix* section.

Access to the GASWorkS support files, such as the Property Tables, is handled by the individual routines that access or manipulate those files. Management routines such as copying and deleting the support files must be performed manually by the User.

A GASWorkS model consists of several files, each possessing the same name but varying extensions. Some of the files represent the header, pipe, node, and customer data associated with each model, while other files represent support or other data files. A full list of the specific file extensions is contained in the *Appendix* section. GASWorkS manages the model files based on the name of the header file.

Only one model may be open at a time. The name of the currently open model is displayed at the top of the main GASWorkS screen. Certain menu items and GDI Commands will prompt for a model name if a function that affects a model file is executed and a model has not already been opened. Once a model has been opened, execution of subsequent model-related routines will apply to the open model.

The routines found in the *File* menu are described in the following sections.

New - *Create A New Model*

The New routine creates a new set of model files. To create a new GASWorkS model...

  ● From the *File* menu, select the *New* item.

  ● The Model Selection screen is displayed. Enter the desired name in the *Model Name* data field, or select an existing name from the *Models* list. Click the *Continue* command button.

● By default, the GDI Window should automatically open. If the GDI Window does not automatically open, select the *View/Edit* item from the *Graphics* menu.

● Use the appropriate GDI Commands to enter the data associated with the new model. As new pipe features and nodes are added, the values initially populating the new items are extracted from the Default Data Values. The Default Data Values and dimensional units may be set by selecting the *Set Defaults* item in the *Utilities* menu.

Notes & Considerations

● If the name of an existing model is selected or entered, a message will be displayed. To overwrite the existing model files, click the *Yes* command button. To return to the Model Selection screen without overwriting the existing model files, click the *No* command button. If you choose to overwrite the model files, the contents of the original files will be sent to the Windows Recycle Bin.

● If the path name is omitted when entering a new name, the current drive and directory (folder) will be assigned as the path. If the extension is omitted, the ".hdr" extension will automatically be applied.

● When executed, GASWorkS will create a new header file and a set of empty data and support files. The contents of the header file will be similar to the last "saved" model.

● See the *Model Selection Screen* topic in the *Data Screens* section for a detailed description of its features.

New Project - *Create A New Model Project*

The New Project routine is similar to the New routine. It creates a new set of model files, but it also allows various settings to be selected before creating the model. To create a new project...

● From the *File* menu, select the *New Project* item.

● The Project Specifications screen is displayed. Enter the name for the new model or select a name, then select the desired support file settings. Select attribute files or background image files to attach as appropriate. Click the *Continue* command button to create the new model.

● By default, the GDI Window should automatically open. If the GDI Window does not automatically open, select the *View/Edit* item from the *Graphics* menu.

● Use the appropriate GDI Commands to enter the data associated with the new model. As new pipe features and nodes are added, the values initially populating the new items are extracted from the Default Data Values. The Default Data Values and dimensional units may be set by selecting the *Set Defaults* item in the *Utilities* menu.

Notes & Considerations

● If the name of an existing model is selected or entered, a message will be displayed. To overwrite the existing model files, click the *Yes* command button. To return to the Model Selection screen without overwriting the existing model files, click the *No* command button. If you choose to overwrite the model files, the contents of the original files will be sent to the Windows Recycle Bin.

● When executed, GASWorkS will create a new header file and a set of empty data and support files.

● If a template model is selected, the contents of the header file and the default values and settings will reflect the values in the selected "template" model. To create a template model, create a new model, change all settings to the desired template values, and save the model to the GASWorkS "...\support\settings" directory (folder).

● See the *Project Specifications Screen* topic in the *Data Screens* section for a detailed description of its features.

Open - *Open An Existing Model*

The Open routine opens an existing model for editing, reporting, or solving. To open an existing model...

● From the *File* menu, select the *Open* item.

● The Model Selection screen is displayed. Enter the desired name in the *Model Name* data field, or select an existing name from the *Models* list. If the file cannot be found, a message will be displayed. When a valid model name has been entered, click the *Continue* command button.

● By default, the GDI Window should automatically open. If the GDI Window does not automatically open, select the *View/Edit* item from the *Graphics* menu.

Notes & Considerations

● If a model is already open, a message may be displayed asking whether to save the changes. Click the *Yes* command button to save changes to the open model, close the model, and open the selected model. Or, click the *No* command button to close the open model without saving changes and open the selected model. Or, click the *Cancel* command button to return to the currently open model.

● If a model from a previous version of GASWorkS is selected, a message will be displayed asking whether to import the file - respond appropriately. If the model is imported, a separate (converted) set of model files will be created with the same name as the original file, followed by a "_10" suffix.

● When a valid model name has been entered, GASWorkS will open and read the associated model files. The various Edit, Report, and Solution routines may be used to manipulate the model.

● See the *Model Selection Screen* topic in the *Data Screens* section for a detailed description of its features.

Close - *Close The Currently Open Model*

The Close routine "closes" the currently open model. To close the currently open model...

       ● From the *File* menu, select the *Close* item.

Notes & Considerations

● If changes have been made to the open model, a message will be displayed asking whether to save the changes - respond appropriately.

● Each time GASWorkS is started, it will automatically open the last open model file. This feature may be disabled by closing the model before exiting GASWorkS.

Save - *Save Model Changes*

The Save routine saves the changes made to the currently open model. To save changes...

       ● From the *File* menu, select the *Save* item.

Notes & Considerations

● If the *Automatically Save At Timed Increments* preference settings option is selected, the model will be saved at specified increments.

● Each time the model is saved, a backup copy of the previous model files is saved. If an existing set of backup files already exist, they are moved to the Windows Recycle Bin.

Save As - *Save Model To Another Name*

The Save As routine saves the changes of the current model to another (different) model name and changes the name of the currently open model to the specified name. To save the current model changes to another name...

       ● From the *File* menu, select the *Save As* item.

B3PE LLC   *Engineering & Software Services*

● The Model Selection screen is displayed. Enter the desired name in the *Model Name* data field, or select an existing name from the *Models* list. Click the *Continue* command button.

● The currently open model will be displayed with the new model name.

Notes & Considerations

● If the name of an existing model is selected or entered, a message will be displayed. To overwrite the existing model files, click the *Yes* command button. To return to the Model Selection screen without overwriting the existing model files, click the *No* command button. If you choose to overwrite the model files, the contents of the original files will be sent to the Windows Recycle Bin.

● When a valid model name has been selected or entered, GASWorkS will copy the contents of the currently open model into the model files associated with the new name.

● The Save As routine saves (copies) the contents of the data stored in memory, not the data stored in the file on disk. To copy the data stored on the disk to another destination, use the Copy routine.

● See the *Model Selection Screen* topic in the *Data Screens* section for a detailed description of its features.

Rename - *Change The Name Of The Currently Open Model*

The Rename routine changes the name of the currently open model to another User-specified name. The name of the various files associated with the model will be changed to the specified name. Unsaved changes to the current model will not be saved. To change the name of the current model...

● From the *File* menu, select the *Rename* item.

● The Model Selection screen is displayed. Enter the desired name in the *Model Name* data field, or select an existing name from the *Files* list. Click the *Continue* command button.

● A message will be displayed when the Rename routine is complete. Click the *OK* command button to clear the message.

Notes & Considerations

● If the name of an existing model is selected or entered, a message will be displayed. To overwrite the existing model files, click the *Yes* command button. To return to the Model Selection screen without overwriting the existing model files, click the *No* command button. If you choose to overwrite the model files, the contents of the original files will be sent to the Windows Recycle Bin.

● When a valid model name has been entered or selected, GASWorkS will change the name of the current model files to the User-specified name.

● The Rename routine changes the name of the data files stored on the disk to a new User-specified name; it does not save the data stored in memory. To save the data stored in memory to another destination, use the Save As routine. The Rename routine does not make a copy of the disk files; it only changes the name of the files. Use the Copy routine to make a copy of the disk files.

● See the *Model Selection Screen* topic in the *Data Screens* section for a detailed description of its features.

Copy - *Copy The Model Files*

GASWorkS provides a routine for copying the files associated with a model. The Copy routine will automatically copy the various model files to a User-specified destination and model name. To copy the files associated with the currently open model...

   ● From the *File* menu, select the *Copy* item.

   ● The Model Selection screen is displayed. Enter the desired name in the *Model Name* data field, or select an existing name from the *Files* list. Click the *Continue* command button.

   ● A message will be displayed when the copy has been completed. Click the *Yes* command button to open the new file or the *No* command button to clear the message.

Notes & Considerations

● If the name of an existing model is selected or entered, a message will be displayed. To overwrite the existing model files, click the *Yes* command button. To return to the Model Selection screen without overwriting the existing model files, click the *No* command button. If you choose to overwrite the model files, the contents of the original files will be sent to the Windows Recycle Bin.

● When a valid model name has been entered or selected, GASWorkS will copy the contents of the current model's files into the destination (target) model files.

● The Copy routine copies the contents of the data stored in the disk file, not the data stored in memory. To save the data stored in memory to another destination, use the Save As routine.

● See the *Model Selection Screen* topic in the *Data Screens* section for a detailed description of its features.

Delete - *Delete The Model Files*

The Delete routine deletes all files associated with a User-selected model, including the backup files. To delete an existing model's files...

● From the *File* menu, select the *Delete* item.

If A Model Is Currently Open -

● A message will be displayed asking whether to delete the currently open model. Either click the *Yes* command button to delete the currently open model, click the *No* command button to return to the Model Selection screen, or click the *Cancel* command button to return to the currently open model.

● If the *Yes* command button is clicked, a message will be displayed asking to confirm the deletion of the selected model. Either click the *Yes* command button to delete the selected model or click the *No* command button to cancel the routine. A message may be displayed when the deletion is complete. Click the *OK* command button to clear the message.

● If the *No* command button is clicked, the Model Selection screen will be re-displayed. Enter or select another name, or click the *Cancel* command button to close the screen.

If No Model Is Currently Open -

● The Model Selection screen is displayed. Enter the desired name in the *Model Name* data field, or select an existing name from the *Models* list. A message will be displayed if the file is not found. When a valid model name has been entered, click the *Continue* command button.

● A message will be displayed asking to confirm the deletion of the selected model. Either click the *Yes* command button to delete the selected model or click the *No* command button to cancel the routine. A message will be displayed when the deletion is complete. Click the *OK* command button to clear the message.

● The Model Selection screen is displayed again. Enter or select another name to delete or click the *Cancel* command button to end the routine.

Notes & Considerations

● When a valid model name has been selected, GASWorkS will "delete" the contents of each file associated with the selected model. The files are not deleted but sent to the Windows Recycle Bin. The files may be manually restored using Windows File Explorer.

● See the *Model Selection Screen* topic in the *Data Screens* section for a detailed description of its features.

Append - *Append The Model Files*

The Append routine provides a means for appending (joining) two models together. All model files are appended except that the header file of the model being appended to is retained as the header file for the new combined model. The model to be appended must be open before attempting to execute the Append routine. To append another model to the currently open model...

● From the *File* menu, select the *Append* item.

● The Model Selection screen is displayed. Enter the desired name in the *Model Name* data field, or select an existing name from the *Models* list. A message will be displayed if a valid file is not found. When a valid model name has been entered, click the *Continue* command button.

● A message will be displayed asking to confirm the appending of the two models. Either click the *Yes* command button to append the selected models or click the *No* command button to cancel the routine.

● Additional messages may be displayed - respond appropriately.

● A message will be displayed when the Append routine is complete. Click the *OK* command button to clear the message. *Note* - If the routine failed or the result was unexpected, close the model without saving the changes.

Notes & Considerations

● Before appending two models, consider renaming or numbering nodes with redundant names between the models; otherwise, redundant names found during the Append routine will automatically be renamed during a sequential number.

● When coincident node locations are encountered, the node in the current model is retained, and the associated pipes in the appending model are snapped to the existing node.

● See the *Model Selection Screen* topic in the *Data Screens* section for a detailed description of its features.

Bundle - *Bundle & Unbundle Model Files*

GASWorkS provides a set of routines for bundling (zipping) and unbundling (unzipping) a model's file set. These routines are described in the following sections.

Bundle A Model - *Zip A Model*

The Bundle routine bundles all of a model's files into a single "zip" format file. This is useful for archiving, backing up, or sharing models with others. The model must be open before it may be bundled. To bundle the files associated with the currently open model...

● From the *Bundle* submenu of the *File* menu, select the *Bundle A Model* item.

● The Bundle Specifications screen is displayed. Select the desired options, then click the *Continue* command button.

● If changes to the model have not been saved, a message will be displayed asking whether to save the model changes before bundling - respond appropriately.

● A message will be displayed stating how many model files were bundled. Click the *OK* command button to clear the message.

Notes & Considerations

● The file created by the Bundle routine will be named the same as the model, except it will have a ".zip" extension. The bundled (zipped) file's format is PKZip 2.04g compatible. The file may be unbundled using the GASWorkS Unbundle routine, or it may be unzipped (extracted) using any compatible file compression program.

● Some firewall and security systems will not allow a zip file containing files with a double extension. The backup files created by GASWorkS contain double extensions. Select the *Exclude Backup Files* bundle option to exclude these files from the bundled file.

● See the *Bundle Specifications Screen* topic in the *Data Screens* section for a detailed description of its features.

Unbundle A Model - *Unzip A Model*

The Unbundle routine unbundles a set of files contained in a single "zip" format file. The zip file may be created either by the GASWorkS Bundle routine or by another zip or compression application. To unbundle a zip file of a model...

● From the *Bundle* submenu of the *File* menu, select the *Unbundle A Model* item.

● The File Selection screen is displayed. Enter the desired name in the *Filename* data field, or select an existing name from the *Files* list. If the name is manually entered, include the ".zip" extension. A message will be displayed if a valid file is not found. When a valid model name has been entered, click the *Continue* command button.

GASWorkS™ 10.1

● If one or more of the model files already exist, a message will be displayed asking whether to overwrite the existing model files. Either click the *Yes* command button to overwrite the file and continue, click the *No* command button to not overwrite the files and continue, or click the *Cancel* command button to cancel the routine.

● When complete, a message will be displayed stating how many model files were unbundled. If unbundled files represent a valid GASWorkS model file set, a message will be displayed asking whether to open the selected model. Either click the *Yes* command button to open the unbundled model or click the *No* command button to clear the message.

Notes & Considerations

● See the *File Selection Screen* topic in the *Data Screens* section for a detailed description of its features.

Purge - *Remove Unrequired Model Files*

The Purge routine will automatically remove any unrequired model files, such as zero-length files and backup files. This process is called "purging" and is useful for cleaning up a model before archiving, backing up, or sharing it with others. The model must be open before it may be purged. To purge the currently open model...

● From the *File* menu, select the *Purge* item.

● A message will be displayed asking to confirm whether to proceed with the Purge routine. Either click the *Yes* command button to purge (delete) the unneeded files from the selected model or click the *No* command button to cancel the routine.

● If the *Yes* command button is clicked, a message may be displayed stating how many files were deleted. Click the *OK* command button to clear the message.

Preferences

GASWorkS allows the User to specify certain User interface behaviors. The available settings are described in the following sections.

Paths - *Set The Default File Locations*

The *Paths* item allows the User to specify the default location of the various model files, initialization files, Property Tables, and support files. These locations specify where to save and retrieve the various files from. The default location is the directory (folder) that GASWorkS uses first when dealing with the various file operations. In most cases, the User may specify other locations when saving or retrieving a file. To specify the preferred locations...

 ● From the *Preferences* submenu of the *File* menu, select the *Paths* item.

 ● The Path Settings screen is displayed. Double-click in the associated data field to display the Path Selection screen and change the path location. Click the *Close* command button to save changes.

Notes & Considerations

● See the *Path Settings Screen* topic in the *Data Screens* section for a detailed description of its features.

Property Tables - *Set The Default Property Tables*

The *Property Tables* item allows the User to specify which Property Tables (contained in the current Property Table path) to use with the current model. To specify the desired Property Tables...

 ● From the *Preferences* submenu of the *File* menu, select the *Property Tables* item.

 ● The Property Table Selection screen is displayed. To change the Property Table path, click the *Change Path Setting* command button. Select the desired location, then select the desired Property Table names. Click the *Close* command button to save changes.

Notes & Considerations

● See the *Property Table Selection Screen* topic in the *Data Screens* section for a detailed description of its features.

Settings - *Set User Interface Options*

The *Settings* item allows the User to specify several settings that affect various behaviors of the GASWorkS User interface. For example, one of these settings causes the GDI Window to automatically open when GASWorkS is first started. To change the preference settings...

- From the *Preferences* submenu of the *File* menu, select the *Settings* item.

- The Preference Settings screen is displayed. Select the desired options. Click the *Close* command button to save changes.

Notes & Considerations

- See the *Preference Settings Screen* topic in the *Data Screens* section for a detailed description of its features.

Save Current Settings

The *Save Current Settings* item allows the current paths and preference settings to be saved for future use. To save the current settings...

- From the *Preferences* submenu of the *File* menu, select the *Save Current Settings* item.

- The File Selection screen is displayed. Enter the desired name in the *Filename* data field, or select an existing name from the *Files* list. Click the *Continue* command button.

- A message may be displayed once the settings have been saved. Click the *OK* command button to clear the message.

Notes & Considerations

- See the *File Selection Screen* topic in the *Data Screens* section for a detailed description of its features.

Open Saved Settings

The *Open Saved Settings* item allows a previously saved preferences file (.pfs) to be opened and applied. To open a saved preferences file...

● From the *Preferences* submenu of the *File* menu, select the *Open Saved Settings* item.

● The File Selection screen is displayed. Enter the desired name in the *Filename* data field or select an existing name from the *Files* list. Click the *Continue* command button.

● A message will be displayed once the settings have been opened and set. Click the *OK* command button to clear the message.

Notes & Considerations

● See the *File Selection Screen* topic in the *Data Screens* section for a detailed description of its features.

Restore A Model - *Restore Model Files*

The *Restore A Model* item provides a routine for restoring the backup copies of a model's data files. GASWorkS automatically creates a backup copy of the various model files each time the Save routine is used. To restore the backup version of the currently open model...

● From the *File* menu, select the *Restore A Model* item.

● A message will be displayed once the restoration is complete. Click the *OK* command button to clear the message.

Notes & Considerations

● GASWorkS exchanges the contents of the original model files and the backup model files. If a model is erroneously restored, immediately re-execute the Restore routine to return the original data. Unless instructed by the User, the current changes will not be saved before the Restore routine is executed.

Restore Samples - *Restore The Sample Files*

When GASWorkS is initially installed, several sample (example) files and models are copied to the destination disk. As the various examples are worked, these files will change. To restore the original (unchanged) example model files...

    ● From the *File* menu, select the *Restore Samples* item.

    ● A message will be displayed when the sample models have been restored. Click the *OK* command button to clear the message.

Quick Export

GASWorkS provides a set of routines for saving the contents of the currently open model to a set of data files in various other formats. These routines are contained in the *Quick Export* submenu of the *File* menu. The Quick Export routines provide a simplified duplication of the functionality of the Export routines.

    ● To export the model data to a selected file format, select the desired format item from the *Quick Export* submenu of the *File* menu.

Notes & Considerations

● More information on the files created using the Quick Export routine may be found in the *Export Routines* section.

Exit GASWorkS - *Closing GASWorkS*

When the *Continue* command button is clicked on the GASWorkS StartUp screen to enter the software, a license of the GASWorkS software will be "checked out."

When exiting the GASWorkS software, there are two options - to "return" the checked-out license or to keep the license "checked out."

    ● To return the "checked out" license, from the *Exit GASWorkS* menu, click the *And Return License* item. In order to return the "checked out" license, the device must be connected to the internet so that the User Count can be updated.

● To keep the license "checked out," from the *Exit GASWorkS* menu, click the *Without Returning License* item. The User Count will <u>not</u> be updated, and the license will remain "checked out" until the license is "returned." This option allows for the software to be used without having access to an internet connection for up to 30 days. Select the *And Return License* item from the *Exit GASWorkS* menu to return a "checked out" license.

Notes & Considerations

● If a model is open when GASWorkS is closed, that model will automatically be reopened the next time GASWorkS is started.

Edit - *Enter & Edit Data*

The majority of the model data editing features are handled in the GDI Window or the various Data Reports. A few special Edit routines are handled outside of those functions. These routines are accessed through the *Edit* menu. The routines found in the *Edit* menu are described in the following sections.

Model Notes - *View/Edit The Model Notes*

To access the Model Notes associated with an existing model...

● From the *Edit* menu, select the *Model Notes* item.

● The Model Notes will be displayed. Edit the Model Notes using standard text editing techniques. Click the *Close* command button to close the screen and save changes.

Notes & Considerations

● The Model Notes can also be viewed from the Header Data Report and on the *Model Notes* data tab on the Solution Data screen.

● See the *Model Notes Screen* topic in the *Data Screens* section for a detailed description of its features.

Mass Update - *Update The Model Data*

The Mass Update routine allows changes en masse to be made to the customer, node, pipe, and associated attribute data based on User-specified criteria. For example, the Mass Update routine could set the *Pipe Efficiency* value to 0.92 for all pipe features where the Pipe Flow Equation is set to Spitzglass-Low Pressure. To make a mass data change...

● From the *Edit* menu, select the *Mass Update* item.

● The Mass Update Specifications screen is displayed. Either enter or select the desired values as appropriate. Click the *Update* command button to perform the update.

Notes & Considerations

● If the *Allow Undo Of Data/Graphic Changes* preference settings option is selected, the original data values may be restored if an erroneous update is performed. To restore the original data values, immediately click the *Restore* command button on the Mass Update Specifications screen.

● See the *Mass Update Specifications Screen* topic in the *Data Screens* section for a detailed description of its features.

Property Tables - *View/Edit The Property Table Report*

GASWorkS uses several Property Tables to maintain common information associated with the various hydraulic feature types. An extensive set of "standard" Property Tables are supplied with GASWorkS. The contents of the standard Property Tables are described in the *Appendix* section. To view or edit the standard Property Tables...

● From the *Edit* menu, select the *Property Tables* item.

● The Property Table Report is displayed. Select the desired data tab (Property Table). View or edit the desired data values by clicking on the associated data cells. Click the appropriate icon to add, delete, or insert a record into the Property Table. Click the *Close* command button.

● If changes have been made, a message will be displayed asking whether to save the changes - respond appropriately.

Notes & Considerations

● See the *Property Table Editor* topic in the *Property Tables* section for a detailed description of its features.

Facility Settings - *View/Edit The Facility Settings*

The *Facility Settings* item provides access to the Facility Settings screen. To view or edit the Facility Settings...

● From the *Edit* menu, select the *Facility Settings* item.

● The Facility Settings screen is displayed. Browse or edit the displayed settings as appropriate. Click the *Apply* command button to close the screen and apply changes.

Notes & Considerations

● The *Save As Default* command button may be used to save the current settings as the default settings to be used when creating a new model.

● See the *Facility Settings Screen* topic in the *Data Screens* section for a detailed description of its features.

Data List Definition - *View/Edit The Data List Definition*

The *Data List Definition* item provides access to the Data List Definition screen. To view or edit a Data List...

● From the *Edit* menu, select the *Data List Definition* item.

● The File Selection screen is displayed. Enter the desired name in the *Filename* data field, or select an existing name from the *Files* list. If a valid file name has not been found, a message will be displayed asking to create the file - respond appropriately. When a valid file name has been entered, click the *Continue* command button.

● The Data List Definition screen is displayed. Make the required changes, then click the *Save* icon to save the changes. When finished, click the *Close* command button to close the screen.

Notes & Considerations

● See the *File Selection Screen* topic in the *Data Screens* section for a detailed description of its features.

● See the *Data List Definition Screen* topic in the *Data Screens* section for a detailed description of its features.

Quick Size Specification - *View/Edit A Quick Pipe Sizing Specification*

The *Quick Size Specification* item provides access to the Quick Pipe Sizing Specification screen. To view or edit a Quick Pipe Sizing Specification...

● From the *Edit* menu, select the *Quick Size Specification* item.

● The File Selection screen is displayed. Enter the desired name in the *Filename* data field, or select an existing name from the *Files* list. If a valid file name has not been found, a message will be displayed asking to create the file - respond appropriately. When a valid file name has been entered, click the *Continue* command button.

● The Quick Pipe Sizing Specification screen is displayed. Make the required changes, then click the *Save* icon to save the changes. When finished, click the *Close* command button to close the screen.

Notes & Considerations

● See the *File Selection Screen* topic in the *Data Screens* section for a detailed description of its features.

● See the *Quick Pipe Sizing Specification Screen* topic in the *Data Screens* section for a detailed description of its features.

Analysis - *Calculating Values*

The GASWorkS Calculation routines are grouped in the *Analysis* menu. The routines in this menu are used to solve a model or calculate gas properties based on a gas composition. The routines found in the *Analysis* menu are described in the following sections.

Solve Single Model

Solving a model involves calculating the unknown *Pressure* and *Flow Rate* values. To solve a GASWorkS model...

● From the *Analysis* menu, select the *Solve Single Model* item.

● The Solution Data screen is displayed. Enter or change the desired data values and options. Click the *Solve* command button to proceed with the solution.

● During the Solution routine, the Solution Log will be displayed. When the Solution routine is complete, browse the contents of the log. Click the *Close* command button to close the log.

Notes & Considerations

● During the solution, a log of the solution activity is written to a file known as a "Solution Log" file. The contents of the log file may be viewed or printed by selecting the *Solution Log* item in the *Report* menu.

● See the *Solution Data Screen* topic in the *Data Screens* section for a detailed description of its features.

Calculate Gas Properties

GASWorkS provides a routine for calculating the gas properties (Specific Gravity, Viscosity, Heating Value, and Specific Heat Ratio) from a gas composition. To use the routine...

● From the *Analysis* menu, select the *Calculate Gas Properties* item.

● The Gas Properties Calculator screen is displayed. Click the *Clear* command button to set all data items to blank (null) values. Enter or select the appropriate values, then click the *Calculate* command button to calculate the gas properties values.

Notes & Considerations

● Some solution options require that a gas properties file be created and saved with the same name as the model. The Gas Properties routine may be used to create the required file.

● The Gas Properties Calculator screen can also be accessed from the Default Data Values screen.

● See the *Gas Properties Calculator Screen* topic in the *Data Screens* section for a detailed description of its features.

GASCalc

If the GASCalc software is installed on the User's machine in its normal location, it may be accessed using the *GASCalc* item from the *Analysis* menu. If GASCalc is not installed or is installed in a non-standard location, the item will be disabled. To open the GASCalc software from within the GASWorkS software...

● From the *Analysis* menu, select the *GASCalc* item.

● If executing the latest version of the GASCalc software, the GASCalc Welcome screen is displayed. Click the *Continue* command button to enter the software. The GASCalc main interface screen is displayed. When finished, exit the GASCalc software.

● If executing the GASCalc 5.0 version of the software, the program will be started, and its main interface screen will be displayed. When finished, exit or minimize the GASCalc software.

GASWorkS™ 10.1 *Menu Items*

Report - *View & Print Data*

The GASWorkS Report routines are grouped in the *Report* menu. The routines located in this menu are used to view and print the data contained in the model and support files. The routines found in the *Report* menu are described in the following sections.

Pipe Data - *View & Edit Pipe Data*

To view the Pipe Data Report, select the *Pipe Data* item from the *Report* menu. The Pipe Data Report will be displayed. Browse the contents of the report. When finished, click the *Close* command button.

Notes & Considerations

● See the *Pipe Data Report* topic in the *Data Screens* section for a detailed description of its features.

Node Data - *View & Edit Node Data*

To view the Node Data Report, select the *Node Data* item from the *Report* menu. The Node Data Report will be displayed. Browse the contents of the report. When finished, click the *Close* command button.

Notes & Considerations

● See the *Node Data Report* topic in the *Data Screens* section for a detailed description of its features.

Customer Data - *View & Edit Customer Data*

To view the Customer Data Report, select the *Customer Data* item from the *Report* menu. The Customer Data Report will be displayed. Browse the contents of the report. When finished, click the *Close* command button.

Notes & Considerations

● See the *Customer Data Report* topic in the *Data Screens* section for a detailed description of its features.

Header Data - *View & Edit Header Data*

To view the Header Data Report, select the *Header Data* item from the *Report* menu. The Header Data Report will be displayed. Browse the contents of the report. When finished, click the *Close* command button.

Notes & Considerations

● See the *Header Data Report* topic in the *Data Screens* section for a detailed description of its features.

System Summary - *Create A System Summary Report*

The System Summary routine provides a choice of certain predefined data and result summaries. To create a System Summary Report...

     ● From the *Report* menu, select the *System Summary* item.

     ● The System Summary Options screen is displayed. Select the desired options to include in the System Summary Report, then click the *Continue* command button.

     ● The System Summary Report will be displayed. Browse the report using the scroll bars. When finished, click the *Close* command button.

Notes & Considerations

● See the *System Summary Options Screen* topic in the *Data Screens* section for a detailed description of its features.

● See the *System Summary Report* topic in the *Data Screens* section for a detailed description of its features.

Data Check - *View The Data Check Report*

The *Data Check* item allows the results of a previously performed Data Check to be viewed. To view the results of a Data Check...

     ● From the *Report* menu, select the *Data Check* item.

     ● The Data Check Report will be displayed. Browse the report using the scroll bars. When finished, click the *Close* command button.

Notes & Considerations

● If the GDI is open, double-click on an error to flag and zoom to the feature associated with the error.

● See the *Data Check Specifications Screen* topic in the *Data Screens* section for a detailed description of its features.

● See the *Data Check Report* topic in the *Data Screens* section for a detailed description of its features.

Solution Log - *View The Solution Log Report*

During the execution of the solution, a log file is created, which contains the activity of the Solution routine. This item is only enabled once the Solution routine has been executed. To view the contents of the Solution Log Report...

> ● From the *Report* menu, select the *Solution Log* item.

> ● The Solution Log Report will be displayed. Browse the report using the scroll bars. When finished, click the *Close* command button.

Notes & Considerations

● If the *Include Exceptions In Solution Log* option is selected, any discovered exceptions will be included in the Solution Log. If the GDI is open, click on a listed exception to flag and zoom to the associated feature.

● See the *Solution Log Report* topic in the *Data Screens* section for a detailed description of its features.

Report Options - *Set The Report Format Options*

GASWorkS allows the User to specify various report format options, including the number of decimal places used when displaying the data and analysis results. These settings are referred to as the Report Options. To set the Report Options...

> ● From the *Report* menu, select the *Report Options* item.

> ● The Report Options screen is displayed. Enter or select the desired settings. Click the *Close* command button to save changes and close the screen.

Notes & Considerations

● See the *Report Options Screen* topic in the *Data Screens* section for a detailed description of its features.

Graphics - *The Graphic Data Interface (GDI) Settings*

The Graphic Data Interface (GDI) allows graphical entry and editing of the various data associated with a model. The GDI provides a map-style (schematic) display of the model and allows point-and-click access to the associated model data. The features associated with the GDI are fully described in the *Graphic Data Interface (GDI)* section.

Various GDI functions are located in the *Graphics* menu. The routines found in the *Graphics* menu are described in the following sections.

View/Edit - *Display The GDI Window & Edit Model Data*

GASWorkS provides graphical access to the model data and schematic through the GDI Window. If the model data was entered graphically, or the nodes have otherwise been assigned coordinates, a schematic of the model will be displayed in the GDI Window. To display the GDI Window...

> ● From the *Graphics* menu, select the *View/Edit* item.

> ● The GDI Window will be displayed. Use the *Scroll* icon to pan the view of the model schematic, or click one of the various "*Zoom*" icons to enlarge or reduce the view of the schematic. Execute one of the GDI Commands to edit and view the model data. To close the GDI Window, click the "*X*" icon in the upper-right corner of the GDI Window.

Settings - *View/Edit The Settings For The GDI Window*

Several settings that control the behavior of the GDI Window are set using the Graphic Settings screen. To access the Graphic Settings screen...

> ● From the *Graphics* menu, select the *Settings* item.

> ● The Graphic Settings screen is displayed. Enter or select the desired settings and options. Click the *Close* command button to save changes and close the screen.

Notes & Considerations

● See the *Graphic Settings Screen* topic in the *Data Screens* section for a detailed description of its features.

Background Settings - *Attach/Edit A Background Image*

Various image formats may be a background to a model's schematic. The assignment and control of the background images are performed using the Background Image Settings routine. To attach, remove, or modify the display of a background image...

● From the *Graphics* menu, select the *Background Settings* item. The Background Image Settings screen is displayed.

To Attach A Background Image -

● Click the *Attach New Image* command button. For the *Attachment File Type*, select the desired file type, then click the *Continue* command button. The File Selection screen is displayed. Enter the desired name in the *Filename* data field, or select an existing Filename from the *Files* list. Click the *Continue* command button.

To Remove A Background Image -

● Select (highlight) the Filename of the background image to be removed from the *Attached Background Image Files* list, then click the *Remove Selected Image* command button.

● Click the *Apply* command button to apply the current settings to the GDI Display immediately.

Notes & Considerations

● See the *Background Image Settings Screen* topic in the *Data Screens* section for a detailed description of its features.

● See the *Attachment File Type Screen* topic in the *Data Screens* section for a detailed description of its features.

Save GDI Image As BMP - *Save The Displayed Image*

The currently displayed GDI Image may be saved to a bitmap (BMP) file for use with other applications. To save the GDI Image to a BMP file...

● From the *Graphics* menu, select the *Save GDI Image As BMP* item.

● The File Selection screen is displayed. Enter the desired name in the *Filename* data field, or select an existing Filename from the *Files* list. Click the *Continue* command button.

● A message will be displayed once the GDI Image has been saved. Click the *OK* command button to clear the message.

Notes & Considerations

● See the *File Selection Screen* topic in the *Data Screens* section for a detailed description of its features.

Copy GDI Image To Clipboard - *Copy The Current GDI Image*

The current GDI Image may be copied to the Windows Clipboard. This allows the GDI Image to be "pasted" into another graphic application. To copy the GDI Image to the clipboard...

● From the *Graphics* menu, select the *Copy GDI Image To Clipboard* item.

● A message will be displayed indicating the result of the Copy routine. Click the *OK* command button to clear the message.

Notes & Considerations

● Use the Windows "paste" command to insert the GDI Image into another application.

● The application into which you intend to paste the image must support the display of graphic images.

Utilities - *Use The Utility Routines*

GASWorkS provides routines for performing various model support functions. These features are referred to as the Utility routines. They include routines for setting default data values and performing various Import, Merge, and Export routines. The Utility routines are grouped in the *Utilities* menu. These routines are described in the following sections.

Set Defaults - *Specify The Default Data Values*

The initial data and unit values used to populate the various data fields are called the "default data." The Set Defaults routine allows the User to set the values for certain data items, which are automatically populated when a new feature is created in the model. To specify the default data values...

● From the *Utilities* menu, select the *Set Defaults* item.

● The Default Data Values screen is displayed. Select the desired data tab, and enter or select the appropriate default data or options. Click the *Close* command button to close the screen and save changes.

Notes & Considerations

● The Set Defaults routine is commonly used to set default data values and dimensional units before entering data. The default data values will be used when a new feature is added. However, the routine can also change the data values for existing features by clicking the appropriate "*Apply...Only*" command button. Be careful using the various "*Apply*" command buttons when features already exist in the model - all associated values will be changed to the specified default data values.

● If the *Use Last Record For Default Values* preference settings option is selected, the "hydraulic" default date values will only used when the first pipe or node feature is entered.

● See the *Default Data Values Screen* topic in the *Data Screens* section for a detailed description of its features.

Customer Data

The *Customer Data* submenu contains several Customer Data Management routines. These routines include the ability to attach an attribute (non-model) database file, update external loads, and manipulate an attached attribute file (database) using Standard Query Language (SQL) commands. These routines are grouped in the *Customer Data* submenu and are described in the following section.

Attribute File - *Attach A Customer Attribute File*

GASWorkS maintains various model-related customer data as part of its model files. It allows non-model data (attribute data) to be contained in a separate database (attribute file) to be attached or connected to the model. The required connection information is specified using the Customer Attribute File Settings screen. To attach (or unattach) a customer attribute file...

● From the *Customer Data* submenu of the *Utilities* menu, select the *Attribute File* item.

● The Customer Attribute File Settings screen is displayed. Enter or select the appropriate data. Click the *Close* command button to save changes and close the screen.

Notes & Considerations

● See the *Attribute File Settings Screen* topic in the *Attribute Data* section for a detailed description of its features.

Update External Loads - *Assign & Update The External Node Loads*

Customer loads are assigned to node loads through the external node load parameter. Sometimes, especially after new data has been imported or changed using the SQL Statement routine, these assignments need to be updated. The Update External Loads routine provides two functions: first, if an attribute file is attached, the routine will update the *Total Load* value for each of the customers, and second, it will reassign the *Total Load* value for each of the customers to the appropriate node's external load value. To update the load assignments...

● From the *Customer Data* submenu of the *Utilities* menu, select the *Update External Loads* item.

● A message may be displayed when the update is complete. Click the *OK* command button to clear the message.

Execute SQL Statement - *Update Attribute Values*

If an appropriate format customer attribute file is attached, values in the attribute file may be changed using an SQL Statement. SQL is an acronym for Standard Query Language, a special "query" language for manipulating and modifying data in a SQL-compatible database. This feature is useful for changing customer load values in the attribute file based on a special relationship or calculation method. To update values in a customer attribute file...

● From the *Customer Data* submenu of the *Utilities* menu, select the *Execute SQL Statement* item.

● The SQL Update Specifications screen is displayed. Enter the desired SQL Statement or select a previously *Saved SQL Statement*, then click the *Execute* command button.

● A message will be displayed when the statement processing is complete. Click the *OK* command button to clear the message.

Notes & Considerations

● See the *SQL Update Specifications Screen* topic in the *Attribute Data* section for a detailed description of its features.

Node Data

The *Node Data* submenu contains several Node Data Management routines. These routines include the ability to attach an attribute (non-model) database file and to manipulate an attached attribute file (database) using Standard Query Language (SQL) commands. These routines are grouped in the *Node Data* submenu and are described in the following section.

Attribute File - *Attach A Node Attribute File*

GASWorkS maintains various model-related node data as part of its model files. It allows non-model data (attribute data) to be contained in a separate database (attribute file) and attached or connected to the model. The required connection information is specified using the Node Attribute File Settings screen. To attach (or unattach) a node attribute file...

● From the *Node Data* submenu of the *Utilities* menu, select the *Attribute File* item.

● The Node Attribute File Settings screen is displayed. Enter or select the appropriate data. Click the *Close* command button to save changes and close the screen.

Notes & Considerations

● See the *Attribute File Settings Screen* topic in the *Attribute Data* section for a detailed description of its features.

Execute SQL Statement - *Update Attribute Values*

If an appropriate format node attribute file is attached, values in the attribute file may be changed using an SQL Statement. SQL is an acronym for Standard Query Language, a special query language for manipulating and modifying data in a SQL-compatible database. This feature is useful for changing node load values contained in the attribute file based on a special relationship or calculation method. To update values in a node attribute file...

● From the *Node Data* submenu of the *Utilities* menu, select the *Execute SQL Statement* item.

● The SQL Update Specifications screen is displayed. Enter the desired SQL Statement or select a previously *Saved SQL Statement*, then click the *Execute* command button.

● A message will be displayed when the statement processing is complete. Click the *OK* command button to clear the message.

Notes & Considerations

● See the *SQL Update Specifications Screen* topic in the *Attribute Data* section for a detailed description of its features.

Pipe Data

The *Pipe Data* submenu contains several Pipe Data Management routines. These routines include the ability to attach an attribute (non-model) database file and to manipulate an attached attribute file (database) using Standard Query Language commands. These routines are grouped in the *Pipe Data* submenu and are described in the following section.

Attribute File - *Attach A Pipe Attribute File*

GASWorkS maintains various model-related pipe data as part of its model files. It allows non-model data (attribute data), to be contained in a separate database (attribute file) and attached or connected to the model. The required connection information is specified using the Pipe Attribute File Settings routine. To attach (or unattach) a pipe attribute file...

● From the *Pipe Data* submenu of the *Utilities* menu, select the *Attribute File* item.

● The Pipe Attribute File Settings screen is displayed. Enter or select the appropriate data. Click the *Close* command button to save changes and close the screen.

Notes & Considerations

● See the *Attribute File Settings Screen* topic in the *Attribute Data* section for a detailed description of its features.

Execute SQL Statement - *Update Attribute Values*

If an appropriate format pipe attribute file is attached, values in the attribute file may be changed using an SQL statement. SQL is an acronym for Standard Query Language, a special query language for manipulating and modifying data in an SQL-compatible database. To update values in a pipe attribute file...

● From the *Pipe Data* submenu of the *Utilities* menu, select the *Execute SQL Statement* item.

● The SQL Update Specifications screen is displayed. Enter the desired SQL Statement or select a previously *Saved SQL Statement*, then click the *Execute* command button.

● A message will be displayed when the statement processing is complete. Click the *OK* command button to clear the message.

Notes & Considerations

● See the *SQL Update Specifications Screen* topic in the *Attribute Data* section for a detailed description of its features.

Import - *Import Data*

GASWorkS supports creating a new model from data saved in various file formats. These routines are controlled through the Import screen. To import a file...

● From the *Utilities* menu, select the *Import* item.

● The Model Selection screen is displayed. Enter the desired name in the *Model Name* data field, then click the *Continue* command button.

● The Import screen is displayed. Select the desired file format from the *File Type* list, then click the *Import* command button.

● The File Selection screen is displayed. Select the desired file to import, then click the *Continue* command button.

● Depending on the specified *File Type*, additional data screens may be displayed to further define the Import Specifications - respond appropriately.

● The *Import Log* on the Import screen will display the status of the Import routine. When finished, click the *Close* command button to exit the screen.

Notes & Considerations

● When a valid model name has been entered, a new model will be created to input the data.

● If the name of an existing model is selected or entered, a message will be displayed. To overwrite the existing model files, click the *Yes* command button. To return to the Model Selection screen without overwriting the existing model files, click the *No* command button. If you choose to overwrite the model files, the contents of the original files will be sent to the Windows Recycle Bin.

● See the *Model Selection Screen* topic in the *Data Screens* section for a detailed description of its features.

● See the *Import* topic in the *Import & Merge Routines* section for a detailed description of its features.

● See the *File Selection* topic in the *Data Screens* section for a detailed description of its features.

Export - *Export Data*

GASWorkS provides various Export routines for sharing its data with other applications through various file formats. These routines are controlled through the Export screen. To export data from the currently open model...

● From the *Utilities* menu, select the *Export* item.

● The Export screen is displayed. Select the desired file format from the *File Type* list, then click the *Export* command button.

● The File Selection screen is displayed. Enter the desired name in the *Filename* data field, then click the *Continue* command button.

● Depending on the specified *File Type*, additional data screens may be displayed to further define the Export Specifications - respond appropriately.

● The *Export Log* on the Export screen will display the status of the Export routine. When finished, click the *Close* command button to exit the screen.

Notes & Considerations

● The model from which the data is exported, must be open before selecting the routine.

● If the name of an existing file is selected or entered, a message will be displayed. To overwrite the existing file, click the *Yes* command button. To return to the File Selection screen without overwriting the existing file, click the *No* command button. If you choose to overwrite the file, the contents of the original file will be deleted.

● See the *Export Screen* topic in the *Export Routines* section for a detailed description of its features.

● See the *File Selection Screen* topic in the *Data Screens* section for a detailed description of its features.

Merge - *Merge Data*

GASWorkS provides a routine for adding data to or updating data in an existing model from data saved in various file formats. These routines are controlled through the Merge screen. To merge data into the currently open model...

    ● From the *Utilities* menu, select the *Merge* item.

    ● The Merge screen is displayed. Select the desired file format from the *File Type* list, then click the *Merge* command button.

    ● The File Selection screen is displayed. Select the desired file to merge, then click the *Continue* command button.

    ● A message will be displayed asking whether to merge the file with the currently open model - respond appropriately.

    ● If the *Yes* command button is clicked, depending on the specified *File Type*, additional data screens may be displayed to further define the Merge Specifications - respond appropriately.

    ● The *Merge Log* on the Merge screen will display the status of the Merge routine. When finished, click the *Close* command button to exit the screen.

Notes & Considerations

● The model into which the data will be merged (added to) must be opened before selecting the routine.

● See the *Merge Screen* topic in the *Import & Merge Routines* section for a detailed description of its features.

● See the *File Selection Screen* topic in the *Data Screens* section for a detailed description of its features.

Purge - *Remove Features*

GASWorkS provides a routine for removing features from an existing model based on a feature list saved in various formats. These routines are controlled through the Purge screen. To purge data from the currently open model...

> ● From the *Utilities* menu, select the *Purge* item.

> ● The Purge screen is displayed. Select the desired file format from the *File Type* list, then click the *Purge* command button.

> ● The File Selection screen is displayed. Select the file that contains the purge (delete) list, then click the *Continue* command button.

> ● A message will be displayed asking whether to continue with the Purge routine - respond appropriately.

> ● If the *Yes* command button is clicked, depending on the specified *File Type*, additional data screens may be displayed to further define the Purge Specifications - respond appropriately.

> ● The *Purge Log* on the Purge screen will display the status of the Purge routine. When finished, click the *Close* command button to exit the screen.

Notes & Considerations

● The model into which the data will be merged (added to) must be opened before selecting the routine.

● See the *Purge Screen* topic in the *Import & Merge Routines* section for a detailed description of its features.

● See the *File Selection Screen* topic in the *Data Screens* section for a detailed description of its features.

Data Check - *Check The Model Data*

GASWorkS provides an extensive Data Check routine. The routine checks for out-of-bounds conditions and general data entry and connectivity errors. The Data Check routine is controlled through the Data Check Specifications screen. To perform a check of the model's data...

> ● From the *Utilities* menu, select the *Data Check* item.

● The Data Check Specifications screen is displayed. Enter or select the desired settings, then click the *Check* command button to proceed with the Data Check.

● When the Data Check is complete, a message will be displayed indicating whether errors were encountered. If errors were found, click the *Yes* command button to view the Data Check Report or click the *No* command button to close the screen without viewing the Data Check Report.

Notes & Considerations

● Once created and closed, the Data Check Report may be viewed at other times by selecting the *Data Check* item from the *Report* menu.

● See the *Data Check Specifications Screen* topic in the *Data Screens* section for a detailed description of its features.

Miscellaneous

Several routines are provided to perform miscellaneous utility functions. These routines are contained in the *Miscellaneous* submenu of the *Utilities* menu. These routines are described in the following sections.

Sort Model

GASWorkS provides various sorting options for reordering the pipes and nodes in the model data files. The sort feature is controlled through the Sort Specification screen. To sort the currently open model's data records...

● From the *Miscellaneous* submenu of the *Utilities* menu, select the *Sort Model* item.

● The Sort Specification screen is displayed. Select the desired sort method from the *Method* list, then click the *Sort* command button.

● A message will be displayed when the sort is complete. Click the *OK* command button to clear the message.

Notes & Considerations

● See the *Sort Specification Screen* topic in the *Data Screens* section for a detailed description of its features.

Window - *Window (Screen) Access Utilities*

Several utilities are provided for arranging and accessing open windows (screens). These routines are grouped in the *Window* menu. These routines are described in the following sections.

Window List - *Access Open Screens*

A list of open screens is displayed in the *Window* menu. When several screens are open and overlapping, it is sometimes difficult to access a specific screen. A screen may be easily and quickly accessed by selecting its name from the Window List. When a screen name is selected, it is moved on top (or in front) of the other screens.

GASWorkS™ 10.1 — *Menu Items*

Help - *View The Help Resources*

Access to various information and documents is provided in the *Help* menu. These routines are described in the following sections.

An internet connection is required to display and view the various GASWorkS help and support documents. Once a document is opened, it can be saved locally if you wish. If the document is saved locally, we suggest storing the documents in the "\app\docs" subfolder of the root (main) GASWorkS 10 program folder.

View Help Guide - *Examples*

GASWorkS provides a Help Guide which has various examples and how-to tips. The Help Guide is designed to complement the documentation contained in this manual - not replace it. To view the Help Guide while connected to the internet, click the *View Help Guide* item from the *Help* menu.

Technical Support - *How To Contact Us*

Several support options are contained in the *Technical Support* submenu. These items are described in the following sections.

Contact Information

To view our contact information, select the *Contact Information* item from the *Technical Support* submenu of the *Help* menu. Our Contact Information will be displayed. Click the *OK* command button to close the screen.

Send Help Request

To send a help request to us, select the *Send Help Request* item from the *Technical Support* submenu of the *Help* menu. If a model is currently open, a message will be displayed asking whether to bundle the current model - respond appropriately. An email will be generated and displayed. Describe the nature of your problem or issue in the email. After completing the problem description and attaching the bundled model, "send" the email. We will reply to the help request at our earliest convenience.

The help request will automatically attach a file containing the *About GASWorkS* information to the email.

Go To Website

To view our website directly, select the *Go To Website* item from the *Technical Support* submenu of the *Help* menu. An internet connection is required for the link to work.

View Getting Started Guide

GASWorkS provides a Getting Started Guide to help with the installation and execution of the GASWorkS software. To view the Getting Started Guide while connected to the internet, click the *View Getting Started Guide* item from the *Help* menu.

View Demonstration Guide

The information contained in the Demonstration Guide is intended to provide an overview of the GASWorkS software by demonstrating a portion of its extensive feature set. The example is for a simple distribution-style piping system; however, please remember that the tasks used to create, solve, and edit this model are applicable to any type of gas piping system. Although the GASWorkS feature set is not fully demonstrated, we hope that the features that are used during the exercises will provide a good appreciation for how GASWorkS may be able to satisfy your network modeling needs.

To view the Demonstration Guide while connected to the internet, click the *View Demonstrated Guide* item from the *Help* menu.

View User's Manual

The User's Manual attempts to provide the User with an understanding of the GASWorkS software by providing extensive descriptions and documentation of its various features. The main body of the User's Manual presents descriptions of the various menu items and data screens. Separate sections are provided for several of the more extensive features. The *Appendix* section presents a primer on network analysis, along with technical documentation of the various equations and data files used and supported by GASWorkS.

To view the User's Manual while connected to the internet, click the *View User's Manual* item from the *Help* menu.

Third Party Terms & Conditions

The GeoMapping functions are supported by services provided by "third parties." Generally, these parties require that the User agrees to certain terms and conditions. Select the *Third Party Terms & Conditions* item to display the terms and conditions associated with any third-party services used in the current GASWorkS session.

About GASWorkS - *Software Information*

Select the *About GASWorkS* item to display the version and setup information about the current GASWorkS software installation. The GASWorkS Information screen is displayed. Once reviewed, click the *OK* command button to close the screen.

DATA SCREENS

The previous section described the function of the GASWorkS menu items. In that section, numerous data and specifications screens were introduced. This section provides a detailed description of those screens. Separate sections are also provided for the more extensive routines and functional groups. The descriptions are listed in alphabetical order based on the screen name.

Attachment File Type

The Attachment File Type screen is accessed by clicking the *Attach New Image* command button from the Background Image Settings screen. The features associated with the screen are described as follows.
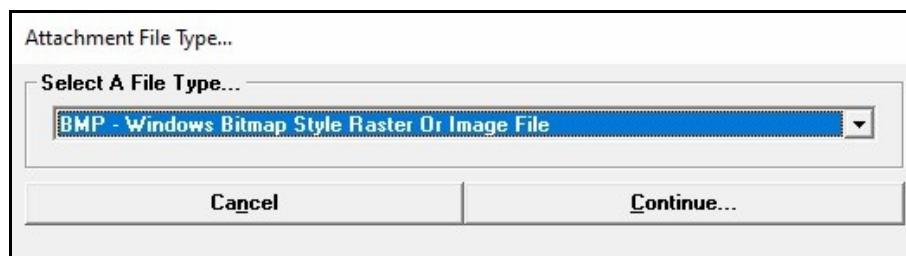
Data Items

Select A File Type - Specifies the image file type.

Command Buttons

Cancel - Closes the screen without saving changes.

Continue - Save the selected image file type and displays the File Selection screen.

Notes & Considerations

● The following image types may be attached as background images to a model:

    .bmp - Windows bitmap style raster or image file
    .dxf - Drawing Exchange File vector style CAD file
    .jpg - Joint Photographic Experts Group (JPEG) style raster or image file
    .kml - Keyhole Markup Language style GIS file
    .pdf - Portable Document Format style document or image file
    .shp - Shapefile style GIS file
    .tif - Tagged Image File Format (TIFF) style raster or image file

● PDF files are handled the same as raster-style files when displayed in the GDI.

GASWorkS™ 10.1   *Data Screens*

Background Image Settings

The settings that control the display of the attached background images are accessed through the Background Image Settings screen. The screen is accessed by either selecting the *Background Settings* item from the *Graphics* menu or selecting the *Background Settings* item from the Data Panel. The features associated with the screen are described as follows and vary depending on the type of image file attached.

Data Items

Attached Background Image Files - Displays the names of the currently attached background image files. Select (highlight) a Filename to make changes to its display properties. The display of a background image file may be controlled by either selecting or unselecting the box adjacent to the Filename in the list.

Command Buttons

Note - Not all command buttons are available for all file types.

Apply - Closes the screen and applies changes.

Attach New Image - Displays the Attachment File Type screen to select a new background image file type to attach to the model.

Calculate Scale Factor - Allows the Scale Factor to be calculated for a raster-style image. When clicked, the Scale Factor Calculation screen is displayed.

Cancel - Closes the screen without saving changes.

Change Path For Selected File - Displays the File Selection screen to change the path for the selected (highlighted) background image file. This item is useful when either the model or background image file or both have been moved.

Close - Closes the screen and saves changes without applying the settings.

Remove Selected Image - Removes the selected (highlighted) background image file from the model.

Select All Images - Selects (highlights) all of the *Attached Background Image Files*.

Select In The Model - Graphically sets the image insertion or origin point.

Turn All Layers On/Off - Allows the display of layers in a DXF-style image to be turned on or off.

Unselect All Images - Unselects all of the *Attached Background Image Files*.

BMP, JPG, & TIF Files

```
┌─ Settings For C:\Program Files (x86)\GASWorkS 10\Files\sample.bmp... ──────────────────┐
│                                                                                        │
│  Scale Factor: │10         │    (Coordinate-To-Pixel Ratio)      │ Calculate Scale Factor │
│                                                                                        │
│  Insertion Point (Upper-Left Coordinates):  X: │0       │   Y: │0       │   │ Select In The Model │
│                                                                                        │
│  Overlap Style: │Combine Images        ▼│                                              │
│                                                                                        │
│                                                                                        │
│                                                                                        │
│                                                                                        │
│  □ Use The Model Path Location For The Background Image File Location                   │
│                                                                                        │
└────────────────────────────────────────────────────────────────────────────────────────┘
```

Data Items

Insertion Point - Specifies the position of the upper-left (northwest) corner of the background image. The X-value represents the horizontal coordinate value of the upper-left corner, and the Y-value represents the vertical coordinate value of the upper-left corner. The values are expressed in the *Coordinates* units.

Overlap Style - Specifies the overlap style used when multiple images are attached. When *Combine Images* is selected, the images will be fused (combined) together to make one image. When *Replace Previous Image* is selected, the current image will lay on top (replace) overlapping portions of the previous image.

Scale Factor - Specifies the number of coordinate units in the model, represented by one pixel in the image. This value sets the scale for the background image. The image size is multiplied by the Scale Factor when the image is processed. The original image data is not changed - the changes are only made to the displayed image. To calculate the value, click the *Calculate Scale Factor* command button. See the following *Notes & Considerations* section for instructions on calculating this value.

Use The Model Path Location For The Background Image File Location - When selected, the background image file is in the same directory (folder) as the associated model. This item is useful when models are frequently shared or moved, eliminating the need to reset the image file location each time the model is moved.

Command Buttons

Calculate Scale Factor - Displays the Scale Factor Calculation screen.

GASWorkS™ 10.1

Data Screens

DXF Files

Data Items

Convert Colors To Gray - When selected, the background image will be displayed using a light gray color.

DXF Layer List - Displays the names of the layers found in the selected DXF background image file. All layers will be listed regardless of their status in the original drawing file. To turn "on" a DXF layer, select the associated layer name. To turn "off" a DXF layer, unselect the associated layer name.

Line Weight - Specifies the width of the lines used to display the background image. The value is expressed in screen pixels.

Origin Shift - Specifies the amount to shift the background image origin. The X-value specifies the amount to move the background image in the east-west (horizontal) direction, and the Y-value specifies the amount to move the background image in the north-south (vertical) direction. The values are expressed in the *Coordinates* units.

Project To Model Coordinates - When selected, the background image will be projected into the associated plane coordinate system.
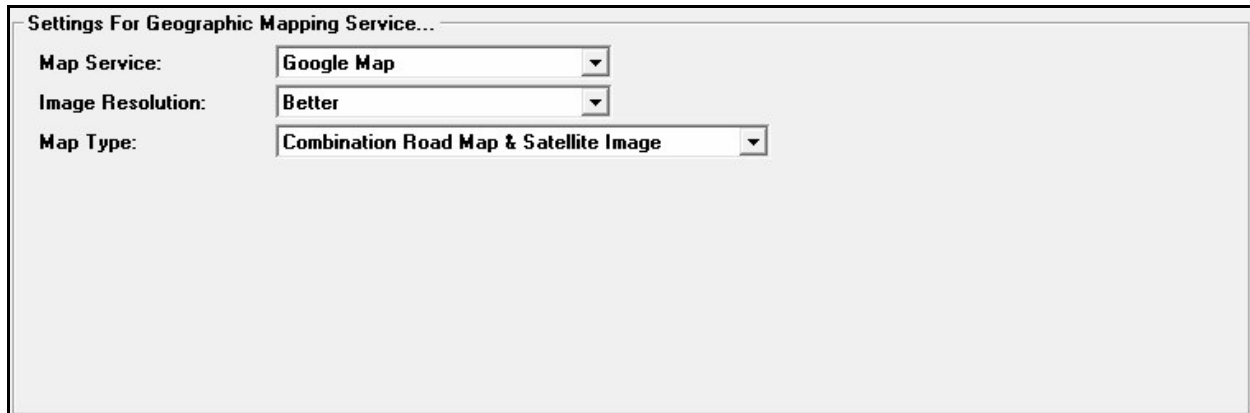
Scale Factor - Specifies the number of coordinate units in the model, represented by one coordinate unit in the drawing. This value sets the scale for the background image. The drawing coordinates are multiplied by the Scale Factor when the image is processed. The original image data is not changed - the changes are only made to the displayed image. See the following *Notes & Considerations* section for instructions on calculating this value.

Use The Model Path Location For The Background Image File Location - When selected, the background image file is located in the same directory (folder) as the associated model. This item is useful when models are frequently shared or moved, eliminating the need to reset the image file location each time the model is moved.

Command Buttons

Turn All Layers On/Off - Turns all of the layers "on" or "off" for the selected (highlighted) DXF background image file. Once all the layers are turned "on" or "off," the settings for the individual layers may be set as desired.

GASWorkS™ 10.1 *Data Screens*

Geographic Mapping Service

```
┌─ Settings For Geographic Mapping Service... ──────────────────────┐
│                                                                    │
│   Map Service:        [ Google Map                        ▼ ]      │
│                                                                    │
│   Image Resolution:   [ Better                            ▼ ]      │
│                                                                    │
│   Map Type:           [ Combination Road Map & Satellite Image ▼ ] │
│                                                                    │
│                                                                    │
│                                                                    │
│                                                                    │
│                                                                    │
│                                                                    │
│                                                                    │
│                                                                    │
│                                                                    │
└────────────────────────────────────────────────────────────────────┘
```

Data Items

Image Resolution - Specifies the resolution of the background image. The options will vary depending on the selected *Map Service*.

Map Service - Specifies the internet-based service that provides the background image.

Map Type - Specifies the type of map to be used for the background image. The options will vary depending on the selected *Map Service*.

KML & SHP Files

Settings For C:\Program Files (x86)\GASWorkS 10\Files\distribution demo_ftg.kml...

Scale Factor (Drawing Unit-To-Coordinate Ratio): **1**

Origin Shift (Model Coordinate Units):   X: **0**   Y: **0**   **Select In The Model**

Line Weight: **1**

Line Style: **Solid**

Point Symbol Size (Default Length Units): **10**

Point Symbol Style: **Circle - Hollow**

Convert Colors To Gray: ☑   Display Color:

☐ Use The Model Path Location For The Background Image File Location

☐ **Project To Model Coordinates**

Data Items

Allow Attribute File Access - For shapefiles (SHP) files, when selected, the data contained in the attribute portion of the shapefile may be displayed in the GDI Image.

Convert Colors To Gray - When selected, the background image will be displayed using a light gray color and not the original colors.

Display Color - Specifies the color used to display the graphic features associated with the image when the *Convert Colors To Gray* option is not selected.

Line Style - Specifies the style to display the lines in the background image. Only used for line (arc) style shapefiles.

Line Weight - Specifies the width of the lines used to display the background image. The value is expressed in screen pixels. Only used for line (arc) style shapefiles.

Origin Shift - Specifies the amount to shift the background image origin. The X-value specifies the amount to move the background image in the east-west (horizontal) direction, and the Y-value specifies the amount to move the background image in the north-south (vertical) direction. The values are expressed in the *Coordinates* units.

Point Symbol Size - Specifies the size used to display the symbols in the background image. The value is expressed in the *Coordinates* units. Only used for point style shapefiles.

Point Symbol Style - Specifies the style of the symbol used to display the point locations in the background image. Only used for point style shapefiles.

Project To Model Coordinates - When selected, the background image will be projected into the associated plane coordinate system.

Scale Factor - Specifies the number of coordinate units in the model, represented by one coordinate unit in the image file. This value sets the scale for the background image. The image coordinates are multiplied by the Scale Factor when the image is processed. The original image data is not changed - the changes are only made to the displayed image. See the following *Notes & Considerations* section for instructions on calculating this value.

Use The Model Path Location For The Background Image File Location - When selected, the background image file is in the same directory (folder) as the associated model. This item is useful when models are frequently shared or moved, eliminating the need to reset the image file location each time the model is moved.

PDF Files

```
⊙ Scale Ratio - 1:   1
○ Scale Factor:              (Coordinate-To-Pixel Ratio)        Calculate Scale Factor

Insertion Point (Upper-Left Coordinates):  X: 0        Y: 0        Select In The Model
Resolution:        150
Overlap Style:   Combine Images          ▼

☐ Use The Model Path Location For The Background Image File Location
```

Data Items

Insertion Point - Specifies the position of the upper-left (northwest) corner of the background image. The X-value represents the horizontal coordinate value of the upper-left corner, and the Y-value represents the vertical coordinate value of the upper-left corner. The values are expressed in the *Coordinates* units.

Overlap Style - Specifies the overlap style used when multiple images are attached. When *Combine Images* is selected, the images will be fused (combined) together to make one image. When *Replace Previous Image* is selected, the current image will lay on top (replace) overlapping portions of the previous image.

Resolution - Specifies the resolution of the background image in dots per inch.

Scale Factor - Specifies the number of coordinate units in the model, represented by one pixel in the converted PDF image. This value sets the scale for the background image. The image size is multiplied by the Scale Factor when the image is processed. The original image data is not changed - the changes are only made to the displayed image. To calculate the value, click the *Calculate Scale Factor* command button (if displayed). See the following *Notes & Considerations* section for instructions on calculating this value.

Scale Ratio - Specifies the scale ratio used to increase (value greater than 1) or decrease (value less than 1) the image size when displayed. A "1:1" Scale Ratio means that one dot (pixel) in the image equals one pixel in the display. This is an alternative method of "scaling" the image.

Use The Model Path Location For The Background Image File Location - When selected, the background image file is in the same directory (folder) as the associated model. This item is useful when models are frequently shared or moved, eliminating the need to reset the image file location each time the model is moved.

Command Buttons

Calculate Scale Factor - Displays the Scale Factor Calculation screen.

Notes & Considerations

● A bitmap (BMP) style image may be used as a background image to a model. The image file must adhere to the standard Windows Bitmap (BMP) specification. A BMP file can contain a scanned image or the raster output of a graphic application that supports the BMP format.

● A drawing exchange format (DXF) style image may be used as a background image to a model. The image file must adhere to the standard drawing exchange format (DXF) file specification. The file can contain the output of a CAD, GIS, or vector-based drawing application that supports the DXF format.

● A joint photographic experts group (JPG/JPEG) style image may be used as a background image to a model. A JPG file is similar to a BMP file in content and application.

● A keyhole markup language (KML) style image may be used as a background image to a model. The image file must adhere to the official KML schema. A KML file can contain graphic (not image) output from Google Earth or another application that supports the KML file format. GASWorkS can only read in KML point or line features.

● A portable document format (PDF) style image may be used as a background image to a model. PDF files may be created by either saving an image or printing to the file with a PDF application. When using a PDF file, it is first converted to a raster-type file in memory and then displayed by the GDI. A PDF is not a raster-type file; however, since it is displayed as a raster-type file, the Scale Factor and other parameters associated with raster-type files are also associated with the PDF file type.

● A shapefile (SHP) style image may be used as a background image to a model. The image file must adhere to the standard shapefile specifications. A SHP file can contain output from a GIS or other application that supports the SHP file format. SHP files contain only one graphic type per file. For example, a SHP file can contain only point-type or line-type features.

● A tagged image file format (TIF) style image may be used as a background image to a model. A TIF file is similar to a BMP file in context and application.

● A file of another format can potentially be used as a background image by first converting the file to a format supported by GASWorkS.

● Multiple files of varying types may be attached to a model.

● When using a raster-style image (BMP, JPG, TIF, PDF), the Scale Factor (Coordinate-To-Pixel Ratio) establishes the "scale" for the images. Raster images are made up of small "dots" called pixels. The scale tells the GDI how many coordinates are represented by one pixel in the file. For example, the number of feet per pixel. To determine this value, divide the original drawing or map scale by the image resolution in dots per inch [dots per centimetre]. For example, the Coordinate-To-Pixel Ratio for a 1 inch = 2000 feet map scanned at 400 dots per inch would be 2000 ÷ 400 = 5 feet/pixel. The Scale Factor may be automatically calculated by clicking the *Calculate Scale Factor* command button for BMP, JPG, and TIF background image files.

● Before attaching a raster-type file, ensure that the *Coordinates* units are set to match the Coordinate-To-Pixel Ratio. For example, if the Coordinate-To-Pixel Ratio is calculated as Feet-per-pixel, the *Coordinates* units should be set to "Feet," or the Coordinate-To-Pixel Ratio should be recalculated to accommodate the current *Coordinates* units.

● Before attaching a DXF file, ensure that the *Coordinates* units are set to match the coordinate values found in the DXF file or use an appropriate Scale Factor to accommodate the *Coordinates* units.

● For SHP, KML, and DXF background image files, the Line Weight and Line Style only apply to files containing line-type features - the Point Symbol Size and Point Symbol Style only apply to files containing point-type features.

● A shapefile (SHP) file in geographic coordinates (1ongitude/1atitude) may be projected into the plane coordinates for display in the GDI. If the SHP file is already in plane coordinates, GASWorkS will first project the SHP file back into longitude/latitude using the associated PRJ file. GASWorkS will then re-project the SHP file using the PRJ file associated with the model. GASWorkS will carry out this projection even if the model is not in projected coordinates as long as a projection file with the same name as the model exists in the model directory (folder).

● KML files are always in geographic (latitude/longitude) coordinates. If the model is in a projected coordinate system, GASWorkS can project the background KML file to model coordinates if a projection file (.prj) is provided.

● When attaching multiple files, the Scale Factor may need to be used to display all of the files in the model Coordinate units.

● When the *Project Coordinates* option is selected, the background image coordinates are first projected into world coordinates (longitude and latitude) based on the values in the associated projection file (.prj). The image is then projected into the appropriate coordinates based on the values in the projection file (.prj) associated with the model.

● See the *Working With The GeoMap Functions* topic in the *Miscellaneous* section for more information on the GeoMap functions. Examples of using the GeoMap commands are found in the GeoMap Examples section of the GASWorkS Help Guide.

Bundle Specifications

A routine is provided that allows the various files associated with a model to be "bundled" (zipped) into a single file. The Bundle routine is controlled by the Bundle Specifications screen. The screen is accessed by selecting the *Bundle A Model* item from the *Bundle* submenu of the *File* menu. The features associated with the screen are described as follows.
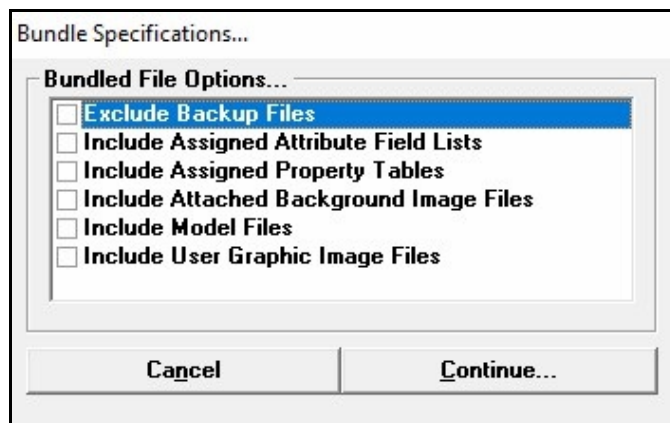
Data Items

Note - The available data items change depending on the options used within the model.

Exclude Backup Files - When selected, the backup files will be excluded in the bundled file.

Include Assigned Attribute Field Lists - When selected, the assigned attribute field lists will be included in the bundled file.

Include Assigned Property Tables - When selected, the currently assigned Property Tables will be included in the bundled file.

Include Attached Attribute Files - When selected, the attached attribute files will be included in the bundled file. This option will only be displayed if a customer, node, or pipe attribute file is attached.

Include Attached Background Image Files - When selected, the attached background image files will be included in the bundled file.

Include Model Files - When selected, the model files will be included in the bundled file.

Include User Graphic Image Files - When selected, the User Graphic Image files will be included in the bundled file.

Command Buttons

Continue - Proceeds with the Bundle routine and creates the bundled file.

Cancel - Closes the screen without creating a bundled file.

Notes & Considerations

● The file created by the Bundle routine will be located in the same directory (folder) as the model, will have the same name as the model, and will end in a ".zip" extension. The bundled file's format is PKZip 2.04g compatible. The file may be unbundled (unzipped) using the *Unbundle A Model* item from the *Bundle* submenu of the *File* menu, or it may be unzipped using any compatible file compression program.

Columns

The User may select the data items (columns) that are displayed in many of the various Reports. The Columns specification screen is used to make these selections. The Columns specification screen is accessed by selecting the *Set Columns* icon from the associated report screen. The features associated with the screen are described as follows.
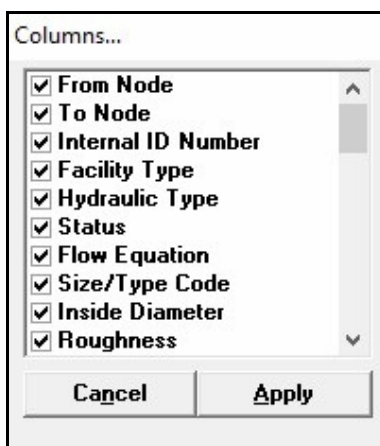
Data Items

Columns - When selected, the associated data item (column) will be displayed in the associated report. The contents of the *Columns* list will vary by report.

Command Buttons

Apply - Closes the screen and applies changes.

Cancel - Closes the screen without applying changes.

Notes & Considerations

● The Set Columns routine may be used to adjust the report width when printing. Normally, the report will contain more information than will fit on a single page. Use the Columns specification screen to eliminate some of the undesired columns from the report by unselecting the unwanted data items (columns).

● To display a specific column, select the column name. To display all of the columns, click the *Restore Columns* icon.

Customer Data Report

The Customer Data Report is accessed by selecting the *Customer Data* item from the *Report* menu. The "report" displays both entered and calculated data values. Entered data values may be changed directly in the report by the User. The features associated with the report are described as follows.

Model Data Tab

Displayed Items

Internal ID Number - Displays the customer's unique (to GASWorkS) internal identification number. This value is automatically assigned and cannot be changed by the User.

Link ID Number - Displays the identification number used to link the customer records in the model to the records in the attribute file or as an identifier for the customer. When no or a "linked" style database is attached, the number may be automatically assigned and cannot be changed by the User. When a "non-linked" style database is attached, the number is assigned by the User. The number should generally represent a unique value in the attribute file (database). The value may be a combination of up to forty (40) alphanumeric characters.

Pipe Facility Type - Displays the assigned Facility Type of the pipe (main) that the customer is attached to. The customer inherits the same Facility Type as the assigned pipe (main).

Record Number - Refers to the position of the current record in the customer data file. An asterisk ("*") next to the Record Number denotes that changes have been made to the record since the last execution of the Solve routine. This value is automatically assigned and cannot be changed by the User.

Supply Main Number - Displays the Link ID Number of the pipe (main) that supplies the customer (the pipe the customer is assigned to). The customer's load will be assigned to the external load of the pipe's From Node or To Node as specified by the *Node Load Application* item. This value cannot be changed directly by the User. *Note -* If the assigned pipe does not have an assigned Link ID Number, the Record Number of the associated pipe will be displayed.

Data Items

Adjust Load - Indicates whether the customer's *Per Unit Load* value is multiplied by the Design Factor when the load is applied to the associated node(s). The *Base Load* value is not changed by this setting.

Load Status - Indicates whether the *Per Unit Load* value is ("on") or is not ("off") applied to the associated nodes.

Load Units - Specifies the dimensional units for the *Per Unit Load* value.

Node Load Application - Specifies how to apply the customer's per unit load.

Per Unit Load - Specifies the load value for each customer "unit." If no attribute file (database) is attached, the item may be manually entered by the User. Otherwise, the item may be automatically assigned depending on how the database is configured. A negative value indicates a demand (flow leaving the system), and a positive value indicates a supply (flow entering the system). The value is expressed in the *Load Units*.

Unit Count - Specifies the total number of units (meters or customers) the selected customer represents. The value must be in the range of 1 to 32,000.

X-Coordinate - Specifies the customer's horizontal location (east-west coordinate).

Y-Coordinate - Specifies the customer's vertical location (north-south coordinate).

Calculated Values

Branch ID - If the selected customer is a "trunk" customer, displays the Internal ID Number of the branch customer assigned to it.

Branch Type - If the selected customer is part of a branch customer, it displays which section the customer is associated with, either "branch" or "trunk."

Edit Date - Displays the date that the customer data was last changed.

Per Unit Adjusted Load - If the *Adjust Load* item is "Yes," displays the *Per Unit Load* value times the Design Factor. Otherwise, displays the *Per Unit Load* value.
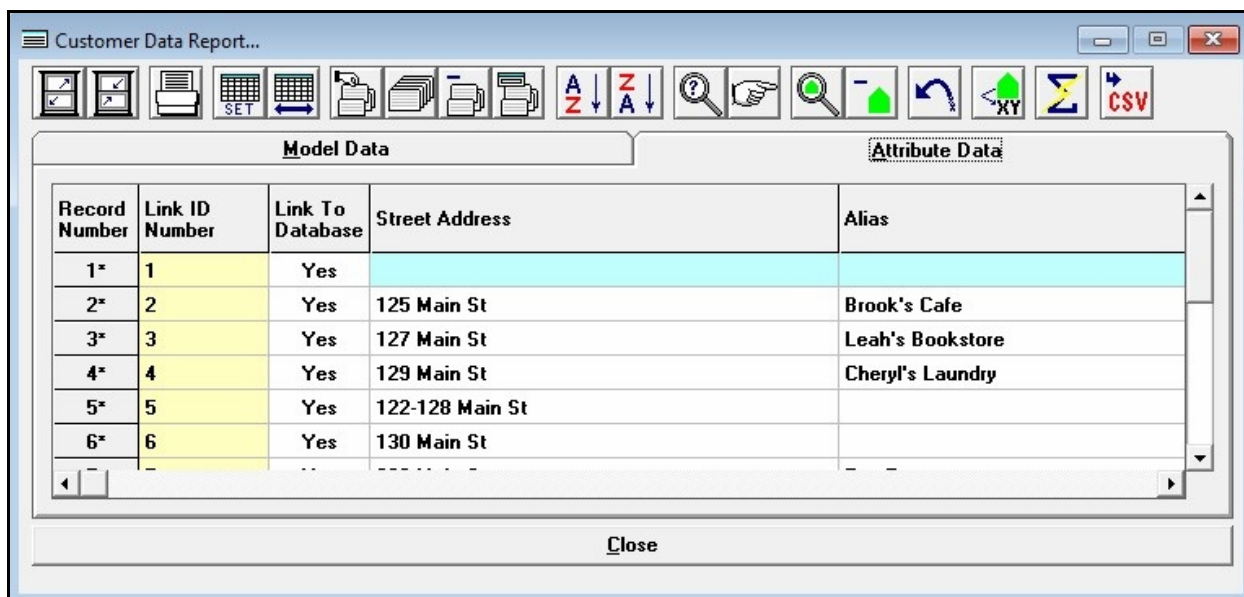
Tap X-Coordinate - Displays the horizontal location (east-west coordinate) of the service tap on the main (pipe) that the associated customer is assigned to. This value cannot be changed directly by the User.

Tap Y-Coordinate - Displays the vertical location (north-south coordinate) of the service tap on the main (pipe) that the associated customer is assigned to. This value cannot be changed directly by the User.

Total Adjusted Load - If the Unit Count is greater than one (1), displays the *Unit Count* value times the *Per Unit Adjusted Load* value.

Attribute Data Tab

If a customer attribute file is attached to the model, the *Attribute Data* tab of the Customer Data Report displays the data in the attribute file. If a database is attached, this data tab will display the values associated with the data fields contained in the associated attribute file (database) and will vary by file. If a "linked" style database is attached, the values may be changed by the User. If a "non-linked" style database is attached, the values are view-only and cannot be changed by the User. The displayed items (columns) will vary depending on the content of the attached file.

Displayed Items

Link ID Number - Specifies the identification number used to link the customer records in the model to the records in an attached attribute file or as an identifier for the customer when no attribute file is attached. When a "linked" style attribute file is attached, the number is automatically assigned and cannot be changed by the User. When a "non-linked" style attribute file is attached, the number may be assigned by the User. In this case, the number should generally represent a unique value in the attribute file. When no attribute file is attached, the item may be used to maintain any User desired alphanumeric value. The value may be a combination of up to forty (40) alphanumeric characters.

Record Number - Refers to the position of the current record in the customer data file. An asterisk ("*") next to the Record Number denotes that changes have been made to the record since the last execution of the Solve routine. This value is automatically assigned and cannot be changed by the User.

Data Items

Link To Database - Indicates whether the customer should be linked to the attribute file (database). If "Yes" is selected, the customer will be linked to the database by matching the Link ID Number assigned in the model to the associated value in the attribute file.

Icons
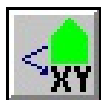
Assign Customer - Allows the selected (highlighted) customer to a location in the GDI Display. When clicked, GDI prompts will appear to select the customer's location and supply main in the GDI Display. This icon only works if the GDI Window is open before the Customer Data Report is opened.

Delete Current Selection - Deletes the current selection set of customer features.
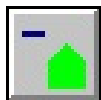
Delete Customer - Deletes the selected (highlighted) customer record.

Edit Selection Set Values - Changes data values en masse for the current selection set of customer features. When clicked, the Multiple Edit Specifications screen is displayed.

Find Customer - Finds and flags the selected (highlighted) customer record in the GDI Display. The icon only works if the GDI Window is open before the Customer Data Report is opened.

Find Record - Finds a customer record in the current selection set by Link ID Number. When clicked, the Find Record specification screen is displayed.

Go To Record - Displays the customer record associated with a specified Record Number at the top of the current data tab. When clicked, the Go To Record specification screen is displayed.

Make Selection Set - Creates a selection set of customers that meet User-specified selection criteria. When clicked, the Query Specifications screen is displayed.
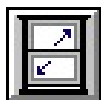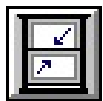
Maximize Screen Size - Increases the report's size to fill the GASWorkS display.

**Print -** Prints the contents of the currently displayed data tab. When clicked, the Print Settings screen is displayed.

Quick Solve - Solves the model using the currently set solution options.

Reset Screen Size - Sets the report's size to its "normal" dimensions and position.

Reset Selection Set - Resets (clears) the current selection set. When clicked, all reportable customer records will be displayed on the associated data tab.
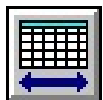
Restore Columns - Restores the display of all data items (columns). When clicked, all of the columns associated with the current data tab will be displayed.

Save To CSV File - Saves the data associated with the current data tab to a comma-separated values file (.csv). When clicked, the file will be created, and a message will be displayed stating the location of the saved file.

Set Columns - Used to specify which columns (data items) are displayed in the Customer Data Report. When clicked, the Columns specification screen is displayed.

Sort Ascending - Sorts the records in ascending order (A to Z or lowest to highest value) based on the Link ID Number.
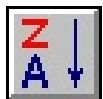
Sort Descending - Sorts the records in descending order (Z to A or highest to lowest value) based on the Link ID Number.
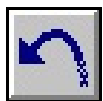
Undo - Restores the most recent data change.

Command Buttons

Close - Closes the Customer Data Report.

Notes & Considerations

● Normally, the report will contain more information than will fit on a single page. If this condition occurs when trying to print, a message will be displayed. When the message is displayed, respond appropriately. To print a report that is too wide for the printer, either eliminate some of the undesired columns using the Set Columns routine or by manually collapsing the undesired columns, set the printer to use *Landscape Orientation*, or reduce the printer font size.

● When a ".csv" file is created, it contains the data and calculated values for records in the current selection set and only for the currently displayed columns. The file may be opened by any application which supports the ".csv" file format.

● Only data items associated with customers attached to pipes assigned Facility Types that are "Active" may be edited in the Customer Data Report. The Facility Settings may be viewed and changed using the *Facility Settings* item from the *Edit* menu.

● If an attribute file is attached, the contents of the file will be displayed on the *Attribute Data* tab of the Customer Data Report. Depending on the type of file attached, the attribute data values may or may not be allowed to be edited by the User.

● The *Node Load Application* options are described as follows:

> *Both Nodes* - Indicates that one-half of the load will be applied to the *External Load* value of the assigned pipe's From Node and To Node.

> *Div...* - Indicates that the load will be applied using the selected diversity load calculation method.

> *From Node* - Indicates that the entire load will be applied to the *External Load* value of the assigned pipes From Node.

> *None* - Indicates that the load is not applied to either node.

> *To Node* - Indicates that the entire load will be applied to the *External Load* value of the assigned pipe's To Node.

● A cell with a "grey" background contains a fixed value. A cell with a "white" background contains a User-entered or selected value. A cell with a "yellow" background contains a calculated value. A cell with a "cyan" background represents an unused or not applicable item. Only values in cells with a "white" background may be changed by the User.

Data Check Report

Errors encountered during the Data Check routine are written to a file known as the "Data Check" file (.dck) or the Data Check Report. The file contains a log of the errors encountered during the Data Check routine. The Data Check Report is accessed by selecting the *Data Check* item from the *Report* menu. The features associated with the report are described as follows.
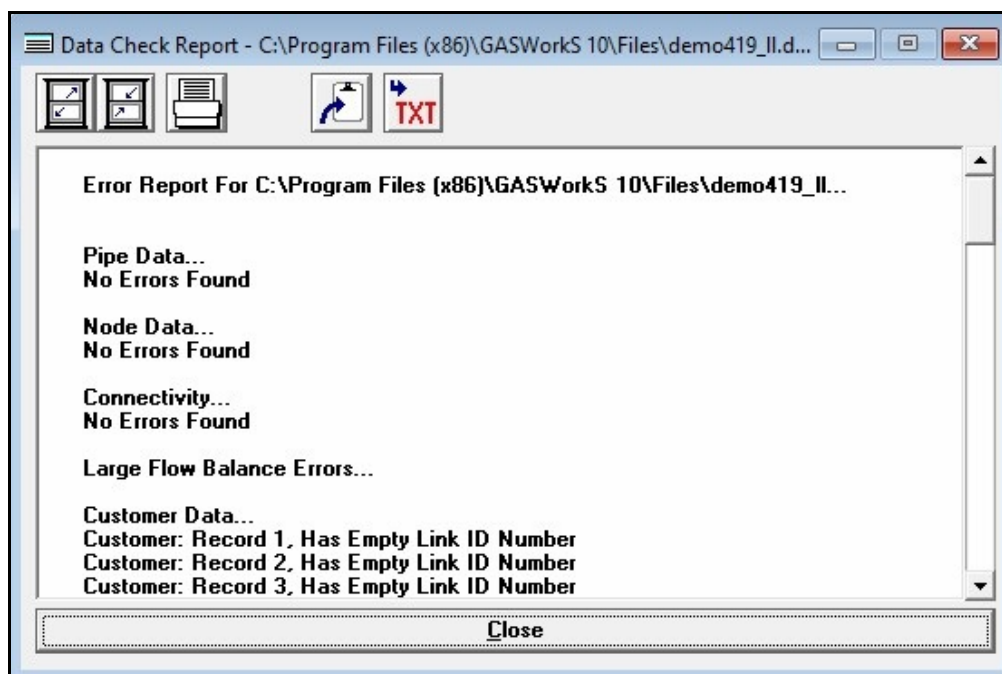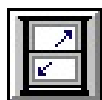

Displayed Items

Error Report - Displays the contents of the Data Check file. To browse the display, adjust the vertical scroll bar adjacent to the display area.

Icons

 Copy To Clipboard - Copies the contents of the Data Check Report to the Windows Clipboard. Once copied, the contents may be pasted into another application.

 Maximize Screen Size - Increases the Data Check Report's size to fill the GASWorkS display.

Print - Prints the contents of the Data Check Report. When clicked, the Print Settings screen is displayed.

Reset Screen Size - Sets the Data Check Report's size to its "normal" dimensions and position.

Save To Text File - Saves the data of the Data Check Report contents to a text file (.txt). When clicked, the File Selection screen is displayed.

Command Buttons

Close - Closes the Data Check Report.

Notes & Considerations

● If the GDI Window is open, double-click on an error to mark it as reviewed and to flag and zoom to the associated feature in the GDI. Marked records become unmarked when the Data Check Report is closed.

● To perform a data check and create a report, select the *Data Check* item from the *Utilities* menu.

GASWorkS™ 10.1   *Data Screens*

Data Check Specifications

The Data Check routine is controlled by the Data Check Specifications screen. The screen is accessed by selecting the *Data Check* item from the *Utilities* menu. The features associated with the screen are described as follows.

Pipe Tab

Data Items

Connectivity - When selected, the model connectivity errors will be included in the Data Check Report.

Diameter - Specifies the minimum and maximum pipe diameter check values.

Efficiency - Specifies the minimum and maximum pipe efficiency check values.

Length - Specifies the minimum and maximum pipe length check values.

Maximum - Specifies the maximum check values for the associated data items.

Minimum - Specifies the minimum check values for the associated data items.

Pipe Data - When selected, the pipe data values will checked, and errors found included in the Data Check Report.

Pipe Property Table - When selected, a data check will be performed to ensure that each Size/Type Code in the pipe data exists in the current Pipe Property Table.

Node Tab

Data Items

Check For Large Flow Balance Errors - When selected, large flow balance errors will be included in the Data Check Report.

Elevation - Specifies the minimum and maximum node elevation check values.

Maximum - Specifies the maximum check values for the associated data items.

Minimum - Specifies the minimum check values for the associated data items.

Node Data - When selected, the node data values will be checked, and any found errors will be included in the Data Check Report.

Pressure - Specifies the minimum and maximum node pressure check values.

Temperature - Specifies the minimum and maximum node (gas flowing) temperature check values.

Total Load - Specifies the minimum and maximum node total load check values. Use only absolute (positive) values. This item refers to the sum of the base and external load values.

Customer Tab

Data Items

Check For Redundant Link ID Numbers - When selected, redundant Link ID Numbers will be included in the Data Check Report.

Check For Redundant Customer Locations - When selected, redundant customer locations will be included in the Data Check Report.

Customer Data - When selected, the customer data values will be checked, and any errors found will be included in the Data Check Report.

Maximum - Specifies the maximum check values for the associated data items.

Minimum - Specifies the minimum check values for the associated data items.

Per Unit Load - Specifies the minimum and maximum load check values for each customer "unit." Use only absolute (positive) values.

Unit Count - Specifies the minimum and maximum total number of units (customers or meters) that each customer feature represents. The *Unit Count* values must be in the range of 1 to 32,000.

Other Tab
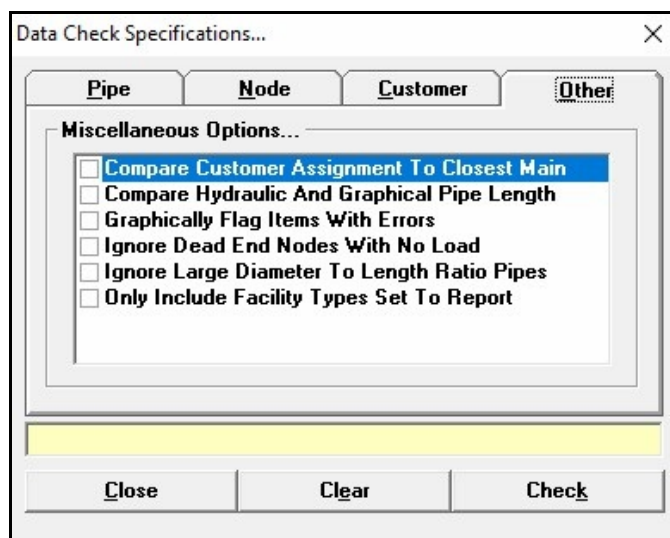
Displayed Items

Status Bar - During processing, displays the status of the Data Check routine.

Data Items

Compare Customer Assignment To Closest Main - When selected, the supply main assigned to the customer will be checked to see if it is the pipe (main) closest to the customer's location.

Compare Hydraulic And Graphical Pipe Length - When selected, the "hydraulic" (entered) length of the pipe will be compared to its graphical length.

Graphically Flag Items With Errors - When selected, flags will be placed on the features that contain errors in the GDI Display.

Ignore Dead End Nodes With No Load - When selected, the "Dead End Nodes With No Load" check will be disabled during the Data Check routine.

Ignore Large Diameter To Length Ratio Pipes - When selected, the "Large Diameter To Length Ratio" check will be disabled during the Data Check routine.

GASWorkS™ 10.1 *Data Screens*

Only Include Facility Types Set To Report - When selected, only the features associated with Facility Types that are set for reporting will be included in the Data Check Report. To change whether a Facility Type is "Report," select the *Facility Settings* item from the *Edit* menu.

Command Buttons

Check - Executes the Data Check routine.

Clear - Sets all data items to blank (null) values.

Close - Closes the screen without executing the Data Check routine.

Notes & Considerations

● When the *Pipe Property Table* option is selected, the Data Check routine will search the current Pipe Property Table for each Size/Type Code (diameter) found in the pipe data file. If the Size/Type Code is not found, an error will be generated. If the Size/Type Code is found, the Data Check routine will check the *Inside Diameter* value to see if it falls within the specified diameter minimum and maximum value limits. If actual *Inside Diameter* values are entered for the Size/Type Code instead of Size/Type Codes, leave the *Pipe Property Table* option unselected.

● The Data Check routine allows the model data to be screened for out-of-bounds, connectivity, and general errors. The routine does <u>not</u> correct the data; it only identifies the errors it encounters and creates a report of its findings. Some of the checks performed are described in the following table:

Connectivity - Reported Errors	
Error Message	**Reported Condition**
Pipes with bad node Record Numbers.	Reports pipes where the assigned node record number exceeds the number of nodes included in the associated node data file. Generally, this occurs when a model has become corrupt during the save or copy process.
Node connected to more than four (4) pipes or more than (6) pipes.	Reports nodes where the number of pipes connected to it is unusually high. Nodes may be connected to from one to up to six pipes. The model will not be able to solve if a node is connected to more than six pipes.

Connectivity - Reported Errors	
Error Message	**Reported Condition**
Node not connected to a pipe.	Reports a node that is not connected to any pipe. Generally, this occurs when a model has become corrupt during the save or copy process.
Node connected to itself.	Reports a node where the pipe From node and To node are the same. Sometimes, it occurs during the Import routine when a pipe length is shorter than the specified Fuzzy Tolerance. It can also occur where a pipe is represented by a circle or closed "loop," for example, on a cul-de-sac end.
Redundant pipes.	Reports two or more pipes that have redundant From and Two nodes. This can occur when pipes are parallel with the same start and endpoints. This can also occur when duplicate pipes have been entered into the model. It often occurs as a result of an Import or Merge routine.
Dead end node with zero (0) load value.	Reports a node that is only connected to one pipe, and the node has a load value assigned to it.

Customer Data - Reported Errors	
Error Message	**Reported Condition**
Missing attribute file (if assigned).	If a customer attribute file is assigned and the file is not found, an error is reported.
Corrupt supply main assignment.	Reports a customer where the record number of the assigned supply main/pipe is larger than the number of pipes contained in the pipe data file. Generally, this occurs when a model has become corrupt during the save or copy process.
Invalid dimensional units.	Reports a customer where the assigned dimensional units for one of the data items do not match the supported dimensional units for that item. Generally, this occurs when an invalid value is assigned during the Import or Merge routine.

Customer Data - Reported Errors	
Error Message	**Reported Condition**
Positive per unit load value.	Reports a customer where the assigned load value is a positive number. Generally, customer load values are assigned a negative number to indicate that the associated value is a demand or gas leaving the system. A positive value would indicate a supply of gas coming into the system.
Out-of-bounds values - Values that exceed the specified maximum value or are less than the specified minimum value.	Reports a customer where one of the assigned values is outside the bounds or limits specified in the check data.
Warning if the load status is set to "off."	Reports a customer where the load status is "off." It's not necessarily an error, just a reminder.
Multiple customers at the same location.	Reports two or more customers located at the same coordinate location. This may indicate redundant customer features caused by an erroneous entry. Or it may occur at locations with multiple meters, which is acceptable.
Invalid Link ID Number.	When a customer attribute file is attached, reports a customer where the value of the assigned Link ID is not found in the customer attribute file.
Assigned main is not the closest to the customer location.	Reports a customer where the assigned supply main/pipe is not the geographically closest to the customer. This could indicate an incorrect assignment.

Node Data - Reported Errors	
Error Message	**Reported Condition**
Missing or empty node data file.	Reports when the associated node data file is empty or missing. This can occur when a model has become corrupt during the save or copy process - especially when the model is manually copied from one location to another.
Duplicate node names.	Reports two or more nodes with identical names or numbers. Although not required, generally, node names should be unique to the model.

GASWorkS™ 10.1 *Data Screens*

Node Data - Reported Errors	
Error Message	**Reported Condition**
Out-of-bounds values - Values that exceed the specified maximum value or are less than the specified minimum value.	Reports a node where one assigned value is outside the bounds or limits specified in the check data.
Both the pressure and loads are either "known" or "unknown."	Reports a node where the "known" settings for the pressure and load values are the same. Except for nodes associated with regulators and compressors, generally, either the pressure or load must be known and the other unknown. The case where both the pressure and load are unknown is not supported.
Invalid dimensional units.	Reports a node where the assigned dimensional units for one of the data items do not match the supported dimensional units for that item. Generally, this occurs when an invalid value is assigned during the Import or Merge routine.
Multiple nodes at the same location.	Reports two or more nodes located at the same coordinate location. This may indicate a connectivity or configuration error.
Warning if the status is set to "off" for the base load or external load.	Reports a node where the status of either the base or external load is "off." It's not necessarily an error, just a reminder.

Pipe Data - Reported Errors	
Error Message	**Reported Condition**
Missing or empty pipe data file.	Reports when the associated pipe data file is empty or missing. This can occur when a model has become corrupt during the save or copy process - especially when the model is manually copied from one location to another.
Missing Property Tables.	Reports when one of the assigned Property Tables is not found.

Pipe Data - Reported Errors	
Error Message	**Reported Condition**
Invalid From Node or To Node Record Numbers.	Reports a pipe where either or both of the record numbers associated with the From and To Node are beyond the number of nodes contained in the node data file. Generally, this occurs when a model has become corrupt during the save or copy process. It may also be reported in the Connectivity check.
Size/Type Code is not found in the Pipe Property Table.	Reports a pipe where the assigned Size/Type Code is not found in the assigned Property Table.
Multiple Size/Type Code matches in the Property Tables.	Reports a pipe where the assigned Size/Type Code is not found multiple times in the assigned Property Table.
Out-of-bounds values - Values that exceed the specified maximum value or are less than the specified minimum value.	Reports a pipe where one assigned value is outside the bounds or limits specified in the check data.
Invalid pipe flow equation.	Reports a pipe where the assigned value for the pipe flow equation is not supported. This can occur when an invalid value has been imported or merged into the model.
Warning if the status is set to "off."	Reports a pipe where the hydraulic status is "off." It's not necessarily an error, just a reminder.
Invalid dimensional units.	Reports a pipe where the assigned dimensional units for one of the data items do not match the supported dimensional units for that item. Generally, this occurs when an invalid value is assigned during the Import or Merge routine.
Large diameter-to-length ratio.	Reports a pipe where the ratio of the diameter to the length is unusually large. This may indicate an error in the diameter or length values. This condition may prevent the solution from becoming complete.
Elevation difference greater than length value.	Reports a pipe where the difference in elevation between the From and To Nodes is greater than the length of the pipe. This condition represents an error in the elevation or length values. The model will not be able to solve this condition.

Pipe Data - Reported Errors	
Error Message	**Reported Condition**
Invalid equivalent length value.	Reports a pipe where the equivalent length of the attached fittings is less than zero. This may occur when an error has occurred while attaching or deleting a fitting from a pipe segment. This can also occur if an invalid equivalent length value is entered into the Fitting Property Table.
Invalid regulator or compressor set pressure values.	Reports a regulator or compressor-style pipe where the set pressure is less than zero.
Unusual "known" or "unknown" settings at the outlet side of regulators and compressors.	Reports a regulator or compressor-style pipe where the status of the "known" and "unknown" settings for the outlet node are unequal. Generally, the pressure and load should be set to "known" for the outlet node of a regulator and the "control" node of a compressor.
Invalid compressor efficiency values.	Reports a compressor-style pipe where the value of the efficiency data item is unusually small or large. Generally indicates an entry error.
Invalid or zero valve opening setting.	Reports a valve-style pipe where the value of the percent open data item is unusually small or large. Generally indicates an entry error. To indicate that the valve is shut off, set the hydraulic status to "off."
Invalid well count, well coefficient, or exponent value.	Reports a well-style pipe where the value of one of the reported data items is unusually small or large. Generally indicates an entry error.
Unsupported *Hydraulic Type*.	Reports a pipe where the assigned hydraulic type is not supported. Generally, this occurs when an invalid value is assigned during the Import or Merge routine.
Invalid or redundant Link ID Numbers.	When a pipe attribute file is attached, it reports a pipe where the value of the Link ID Number is not found in the assigned attribute file or the value is found to be assigned to multiple pipes in the model.

Pipe Data - Reported Errors	
Error Message	**Reported Condition**
Difference in graphic and hydraulic length.	Reports a pipe where the assigned hydraulic length differs from the calculated graphic length. This can occur when a pipe has been created by importing or merging data, and the length value is assigned to a value from the imported file. It's not necessarily an error, just a reminder to check it.

Data List Definition

The Attribute Data and Sizing Group are examples of items supported by data lists. The data lists may be modified by the User using the Data List Definition screen. To create or modify a data list, select the *Data List Definition* item from the *Edit* menu. The features associated with the screen are described as follows.

Displayed Items

Item # - Displays the position that the "item" will be displayed in the data list. This is internally assigned and cannot be changed by the User.

Data Items

Item Description - Specifies the description for the "item" displayed in the data list.

Item Index - Specifies an index value associated with the list item. If the data list is assigned to an integer type field in the Attribute Database Definition, the index value may indicate the assigned "list" item. The use of an index value is optional; however, if used, the value should generally be unique to the associated data list. The index value must be an integer value. If "Remark" is selected as the value, the *Item Description* will not be listed when the Data List is displayed.

Item Value - If displayed, specifies a numeric value to be associated with the "list" item. The value may be an integer or floating point number.
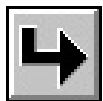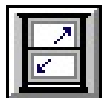
GASWorkS™ 10.1 *Data Screens*

Icons
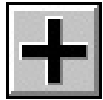
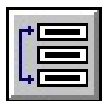**Add Item -** Adds a new item to the end of the definition.
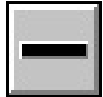
Delete Item - Deletes the selected (highlighted) item.

Insert Item - Inserts a new item just before the selected (highlighted) item.

Maximize Screen Size - Increases the Data List Definition screen size to fill the GASWorkS display.

Move Item - Moves the selected (highlighted) item to a new location in the Data List Definition. When clicked, a message will be displayed to select the new location. Click the *OK* command button to clear the message, then select the desired location.
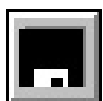
Open - Opens a previously saved Data List Definition file (.lst). When clicked, the File Selection screen is displayed.

Print - Prints the contents of the Data List Definition. When clicked, the Print Settings screen is displayed.
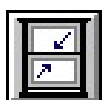
Reset Screen Size - Sets the Data List Definition screen size to its "normal" dimensions and position.

Save - Saves the contents of the currently displayed Data List Definition. When clicked, the File Selection screen is displayed. Once saved, the file may be opened and used with other models.

Command Buttons

Close - Closes the screen. A message will be displayed if changes have been made without saving - respond appropriately.

GASWorkS™ 10.1

Data Screens

Notes & Considerations

● The *Item Index* value must be an integer value between -32000 and 32000, including 0 (zero).

● The *Item Value* item may or may not be displayed. The value is used by certain calculation routines. If the item is displayed, its value may be any double-precision numeric value.

● Several predefined data lists are included when the software is installed. The data lists are normally installed in the GASWorkS Support directory (folder) (...\support\datalists).

Default Data Values

The default data values are used to initially populate the pipe and node data values when creating a new model or adding new features to an existing model. These values are set using the Default Data Values screen. The screen may be accessed by selecting the *Set Defaults* item from the *Utilities* menu or clicking the *Set Default Data Values* command button on either the Import or Merge screen. The features associated with the screen are described as follows.

Dimensional Units Tab

Data Items

Note - Some dimensional units are common throughout the model; some may be set individually. Units that may be set individually are noted as "default" in the following.

Coordinates - Specifies the dimensional units to use for the model coordinate values.

Customer Load - Specifies the "default" dimensional units for the customer load values.

Diameter - Specifies the "default" dimensional units for the diameter values.

Efficiency - Specifies the dimensional units for the efficiency values.

Elevation - Specifies the dimensional units for the elevation values.

Heating Value - Specifies the dimensional units for the heating values.

Length - Specifies the default dimensional units for the length values.

Node Load - Specifies the default dimensional units for the node (base) load values.

Pipe Flow Rate - Specifies the default dimensional units for the flow rate and capacity values.

Pressure - Specifies the default dimensional units for the node pressures and pipe pressure (drop) values.

Temperature - Specifies the dimensional units for the temperature values.

Velocity - Specifies the dimensional units for the velocity values.

Viscosity - Specifies the dimensional units for the absolute or dynamic viscosity values.

GASWorkS™ 10.1   *Data Screens*

Hydraulic Data Tab

Data Items

Allow Pipe Sizing - Specifies the default setting for the *Allow Pipe Sizing* option.

Node Base Load - Specifies the default node base load value. A negative value indicates a demand (flow leaving the system), and a positive value indicates a supply (flow entering the system). If customer features will be used with the model, generally, set the default *Node Base Load* value to zero (0).

Node Elevation - Specifies the default node elevation value.

Node Pressure - Specifies the default node pressure value. In general, this value should be set to zero (0).

Node Temperature - Specifies the default node temperature (gas flowing temperature) value.

Pipe Efficiency - Specifies the default pipe efficiency (hydraulic efficiency) value.

Pipe Equation - Specifies the default pipe flow equation.

Pipe Size/Type - Specifies the default pipe Size/Type Code (*Diameter* value).

Value Is Unknown - Specifies the default "known" or "unknown" setting for the associated data value.

Gas Properties Tab

Data Items

Heating Value - Specifies the default heating value (energy content).

Specific Gravity - Specifies the default specific gravity value.

Specific Heat Ratio - Specifies the default specific heat ratio value.

Viscosity - Specifies the default viscosity value.

Value Is Unknown - Specifies the default "known" or "unknown" setting for the gas property values.

Miscellaneous Tab

Data Items

Node Increment - Specifies the default node increment value.

Pipe Facility Type - Specifies the default pipe Facility Type.

Pipe Sizing Group - Specifies the default pipe Sizing Group.

Command Buttons

Apply...Only - Applies the default data values associated with the currently displayed data tab to <u>all</u> pipes and nodes in the currently open model. This routine is useful when only a specific set of the default data values needs to be changed (for example, to reset the dimensional units).

Apply All Values - Applies <u>all</u> of the default data values and settings to <u>all</u> pipes and nodes in the currently open model. This routine is useful when a new model is created using the wrong default data values and settings.

Calculate Gas Properties From Composition - Calculates the gas properties from a known composition. The command button is displayed on the *Gas Properties* data tab.

Cancel - Closes the screen without saving changes.

Close - Closes the screen and saves changes.

Open Settings - Opens a previously saved default settings file (.dfs).
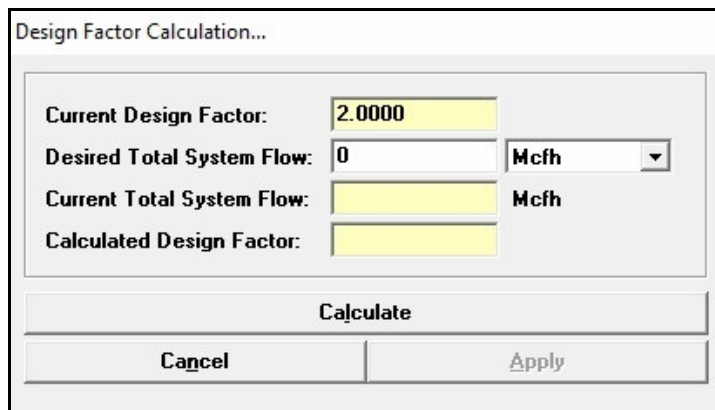
Save Settings - Saves the current default data values and settings to a file.

Notes & Considerations

● Depending on the value of the *Use Last Record For Default Values* preference settings option, the default data values are used to initially populate the pipe and node data items when a new feature is added to the model. These include selected default data values that are most commonly used for the system being modeled. These values and settings may need to be changed occasionally, depending on the nature of the current model.

● When the *Use Last Record For Default Values* preference settings option is selected, the default pipe and node values are only used for the first record, and thereafter, the data from the last record in the model will then be used.

● Any default data values may be overridden as the new features are entered (if the *Allow Data Entry During New Feature Entry* graphic settings option is selected) or after the data has been entered using the data editing routines.

Design Factor Calculation

The Design Factor Calculation screen may be accessed by clicking the *Calculate* command button on the *Solution Data* tab of the Solution Data screen or by selecting the *Calculate Design Factor* item from the Analysis menu. The features associated with the screen are described as follows.

Data Items

Desired Total System Flow - Specifies the desired total system flow value.

Calculated Items

Calculated Design Factor - Displays the calculated Design Factor value, which, when used, would cause the total system flow to equal the entered value.

Current Design Factor - Displays the current value of the Design Factor. The item is automatically populated and cannot be directly changed by the User.

Current Total System Flow - Displays the current value of the total system flow. The item is automatically populated and cannot be directly changed by the User.

Command Buttons

Apply - Closes the calculation screen and applies the calculated Design Factor value to the Solution Data screen.

Calculate - Calculates the associated Design Factor.

Cancel - Closes the calculation screen without applying the calculated Design Factor.

Notes & Considerations

● The Design Factor adjusts the node and customer load values during the Solution routine. The value may be used to raise or lower the total demand on the model.

● The Total System Flow is the summation of all the flows into the model.

● Adjustment to individual node or customer features may be controlled using either the *Adjust Base Load* or *Adjust External Load* options associated with a node and/or the *Adjust Load* option associated with a customer.

● For node features, only the loads for nodes that are associated with a pipe Facility Type set to "solve" are adjusted.

Facility Settings

The Facility Settings associated with a model are managed using the Facility Settings screen. The Facility Settings screen may be accessed by selecting the *Facility Settings* item from the *Edit* menu. The features associated with the screen are described as follows.

Record Number	Index No	Facility Type	Display
1	0	UnSpecified	X
2	5	Main - Low Pressure	X
3	6	Main - Medium Pressure	X
4	7	Main - Intermediate Pressure	X
5	8	Main - High Pressure	X
6	9	Main - Proposed	X

Displayed Items

Record Number - Refers to the position of the current Facility Type in the facility settings file. This value is automatically assigned and cannot be changed by the User.

Data Items

Active - Indicates whether the data features with the associated Facility Type may be changed using the various Edit routines. Generally, all features associated with an "inactive" Facility Type are excluded from the Edit routines. The exception is that if the *Ignore Facility Settings On Mass Update* preference settings option is selected, all records will be included in changes made by the Mass Update routine.

Display - Indicates whether the data features with the associated Facility Type will be displayed in the GDI Display.

Facility Type - Specifies the name of the facility.

Index No (Number) **-** Specifies the value of the "index" of the associated Facility Type. Facility Types are stored as a number value - the index number represents that value. This value must be unique to the model.

Pipe Flow Equation - Specifies which pipe flow equation to use when a pipe associated with the Facility Type has the *Flow Equation* in the Pipe Data set to "Facility Table."

Pipe Line Color - Specifies which color to use when a pipe associated with the Facility Type has the *Pipe Line Color* in the Pipe Data set to "Facility Table."

Pipe Line Style - Specifies which line style to use when a pipe associated with the Facility Type has the *Pipe Line Style* in the Pipe Data set to "Facility Table."

Pipe Line Thickness - Specifies which line thickness to use when a pipe associated with the Facility Type has the *Pipe Line Thickness* in the Pipe Data set to "Facility Table."

Pressure Units - Specifies the pressure units to use when a pipe associated with the Facility Type has the *Pressure Units* in the Pipe Data set to "Facility Table." This parameter is included for future implementation; it is not currently used.

Report - Indicates whether the data features associated with the Facility Type will be included in the various Report routines.

Solve - Indicates whether the data features associated with the Facility Type are included in the Solution routine. *Note -* Be careful when excluding features from the Solution routine, as a non-solvable configuration may result when excluding portions of the piping system.

Icons
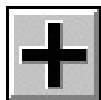
Add Record - Adds a new record to the end of the Facility Settings table.
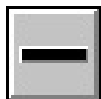
Delete Record - Deletes the selected (highlighted) record.

Insert Record - Inserts a new record just before the selected (highlighted) record.
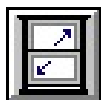
Maximize Screen Size - Increases the Facility Settings screens size to fill the GASWorkS display.
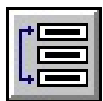
Move Record - Moves the selected (highlighted) record to a new location in the Facility Settings. When clicked, a message will be displayed to select the new location. Click the *OK* command button to clear the message, then select the desired location.

Open - Opens a previously saved Facility Settings file (.fac). When clicked, the File Selection screen is displayed.

Print - Prints the contents of the Facility Settings table. When clicked, the Print Settings screen is displayed.
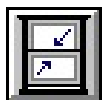
Reset Screen Size - Sets the Facility Settings screen's size to its "normal" dimensions and position.
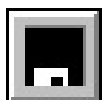
Save - Saves the contents of the currently displayed Facility Settings. When clicked, the File Selection screen is displayed. Once saved, the file may be opened and used with other models.

Command Buttons

Apply - Closes the screen and applies changes.

Cancel - Closes the screen without saving changes.

Save As Default - Saves the current facility settings as the default settings to be used when creating a new model.

Notes & Considerations

● The Facility Settings screen lists the Facility Types associated with the currently open model or currently opened Facility Settings file and the associated settings. The settings control how the data features associated with a specific Facility Type are handled by the various Report, Display, Edit, and Solution routines.

● Facility Types are primarily assigned to pipe data features. The node and customer features inherit the Facility Type of the pipe they are associated with or assigned to.

● Nodes may be associated with several pipe features, and each pipe could possess different Facility Types and settings. In these cases, the node handling will vary according to the routine. If at least one associated pipe possesses a Facility Type that allows a process, then the node is allowed in any associated activities. For example, if at least one pipe is set to "Display," the node is displayed - if at least one pipe is set to "Report," the node is reported, etc.

● When a Facility Type is assigned to a feature, the Index No for the Facility Type is saved with the feature's data. Several "special" Facility Types have pre-assigned Index No's, these begin at 30000. (For example, there is a special Facility Type for the Color Legend.) The Index No for all other Facility Types may be entered by the User. When manually entering an Index No, ensure the number is unique and between 1 and 29999.

File Selection

The File Selection screen is used for accessing various support and other files. The File Selection screen is displayed by various routines. The features associated with the screen are described as follows.
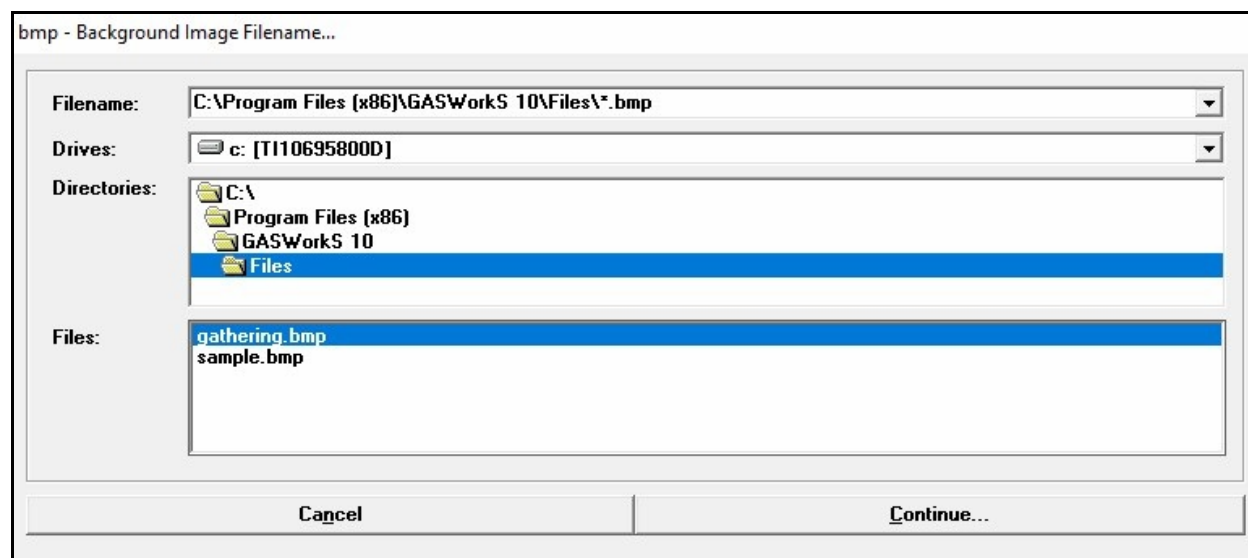
Data Items

Directories - Specifies the directory (folder) on the selected disk drive. When a directory has been selected, any files matching the specified extension or search pattern will be displayed in the *Files* panel.

Drives - Specifies the disk drive to use. When a drive is selected, the directories contained on that drive will be listed in the *Directories* panel.

Filename - Specifies the name of the file to be used. A list of the most previously used files may be accessed from the drop-down list. The Filename may be entered by various means. If the Filename is manually entered, include the path. Extension values will vary according to the file type.

Files - Specifies the matching files in the selected directory on the disk drive.

Command Buttons

Cancel - Closes the screen without saving changes and without preceding the routine that invoked the screen.

Continue - Accepts the displayed Filename and proceeds with the associated routine.
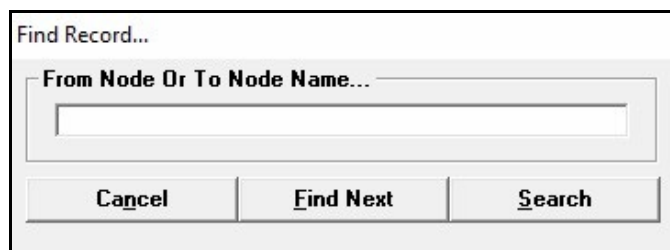
Notes & Considerations

● Depending on the routine that invoked the File Selection screen, the response to errors that may occur while selecting a file will vary. If the routine requires a new Filename and an existing Filename is selected, the screen will prompt whether the file should be overwritten - respond appropriately. If an existing file is chosen to be "overwritten," the file is not destroyed; it is sent to the Windows Recycle Bin. If the routine requires the name of an existing file and the name of a non-existent file is specified, the screen will respond with either a message that the file does not exist or will prompt whether the file should be created depending on the nature of the calling routine - respond appropriately.

● Wild card characters may be entered in the *Filename* data field to alter the contents of the *Files* panel. Valid characters are "*" for multi-character replacement and "?" for single-character replacement.

● *Drives*, *Directories*, and *Filename* values must correspond to the standard "DOS" format. That is, drive letters must be followed by a colon ":" directories must be separated by a backslash "\," and extensions must be proceeded by a dot ".".

● When manually entering a value in the *Filename* data field, include the drive, full path name, Filename, and extension values.

Find Record

The Find Record routine is used by many of the report screens. It allows records to be located by searching for a specified data value. The search criteria are entered using the Find Record specification screen. The Find Record specification screen is accessed by clicking the *Find Record* icon from the associated report screen. The features associated with the screen are described as follows.

```
Find Record...
┌─ From Node Or To Node Name... ─────────────┐
│  [                                        ] │
└────────────────────────────────────────────┘
   Cancel        Find Next        Search
```

Data Items

Search Value - Specifies the data value to search for. The title of the data field and the meaning of the value will vary depending on which report was used to open the screen.

Command Buttons

Cancel - Closes the screen without performing the search.

Find Next - If enabled, searches for another occurrence of the most previously specified search value.

Search - Initiates the search.

Notes & Considerations

● The *Search Value* will vary depending on the report that invoked the Find Record routine. For pipe data, both the From Node and To Node Names are searched. The node *Name* values are searched for node data. The *Link ID Number* values are searched for customer data.

● When executed the first time, searching begins from the start of the file or current selection set for a record matching the search criteria. If a match is found, the matching record is made current. To find other occurrences, re-execute the Find Record routine and click the *Find Next* command button. If no match is found, a message will be displayed.

● When the *Find Next* command button is clicked, the search begins at the last matching record and then searches to the end of the file or selection set. If no match is found, the search begins again at the beginning of the file or selection set and continues until a match is found or the entire file or selection set has been searched.

Fitting Data

The Fitting Data screen is accessed by clicking in an *Attached Fittings* cell on the *Hydraulic Data* tab of the Pipe Data Report. The features associated with the screen are described as follows.

```
Fitting Data - Rec Number: 1, ID Number: 1, Nodes: Reg_Out to 10
┌─ Fitting Data... ─────────────────────────────────────────────┐
│  Available Fittings:   [1/2 L-45 Thd                      ▼]  │
│                                                                │
│  Attached Fittings:   ┌──────────────────────────────────┐    │
│                       │                                  │    │
│                       │                                  │    │
│                       │                                  │    │
│                       └──────────────────────────────────┘    │
│                                                                │
│  Total Equivalent Length: [          ]   Feet                 │
│                                                                │
│  [    Recalculate Length    ]  [ Clear Attached Fittings List ]│
└────────────────────────────────────────────────────────────────┘
│  [        Cancel        ]      [         Close          ]      │
```

Data Items

Attached Fittings - Displays the attached fittings for the associated pipe segment.

Available Fittings - Specifies the fittings available to attach to the associated pipe segment.

Calculated Values

Total Equivalent Length - Displays the total equivalent length of the attached fittings in the *Attached Fittings* list. This value is added to the pipe's hydraulic length during the Solution routine.

Command Buttons

Cancel - Closes the screen without saving changes.

Clear Attached Fittings List - Removes all of the fittings from the *Attached Fittings* list.

Close - Closes the screen and save changes.

Recalculate Length - Recalculates the *Total Equivalent Length* value when another fitting is attached.

Notes & Considerations

● To attach (add) a fitting, select an item from the *Available Fittings* list.

● To remove a fitting, double-click on the associated item in the *Attached Fittings* list.

● The equivalent length of any attached fitting is added to the pipe hydraulic length when performing hydraulic calculations during the Solution routine.

● The *Available Fittings* list is populated with the values from the currently assigned Fitting Property Table.

● The total equivalent length of the attached fittings is calculated based on the inside *Diameter* value of the associated pipe.

Gas Properties Calculator

GASWorkS provides a routine for calculating the Heating Value, Specific Gravity, Specific Heat Ratio, and Viscosity values based on an entered gas composition. The routine may be accessed by selecting the *Calculate Gas Properties* item from the *Analysis* menu or clicking the *Calculate Gas Properties From Composition* command button from the Default Data Values screen. The features associated with the screen are described as follows.

Data Items

Gas Composition - Specifies the individual components found in the gas mixture. The values are expressed as mole percentages. The sum of all of the percentages must equal 100%. If a component does not exist in the gas, leave its data field empty or set its value to zero (0).

Heating Value Method - Specifies the method used to calculate the heating value. The gas properties will need to be recalculated if the method is changed.

Engineering & Software Services

GASWorkS™ 10.1   *Data Screens*

Calculated Values

Heating - Displays the calculated (gross) heating value associated with the entered composition.

Specific Gravity - Displays the calculated specific gravity value associated with the entered composition. The value represents the ratio of the weight of the gas composition compared to dry air.

Specific Heat Ratio - Displays the calculated ratio of specific heat values associated with the entered composition. The value represents the ratio of the constant pressure-specific heat to the constant volume-specific heat.

Viscosity - Displays the calculated (absolute or dynamic) viscosity value associated with the entered composition.

Command Buttons

Apply - The function of this button varies depending on the source that opened the screen. If the screen is opened from the *Analysis* menu, applies the calculated gas properties values to all of the nodes in the current model. If the screen was opened from the Default Data Values screen, applies the calculated values to the default gas properties values.

Calculate - Calculates the various gas property values from the entered composition.

Clear - Sets all data items to blank (null) values.

Close - Closes the screen and saves changes.

Open - Opens a previously saved Gas Properties file (.prp). When clicked, the File Selection screen is displayed.

Print - Prints the data values and results. When clicked, the Print Settings screen is displayed.

Save - Saves the contents of the calculation screen. When clicked, the File Selection screen is displayed. Once saved, the file may be opened and used with other models.

Notes & Considerations
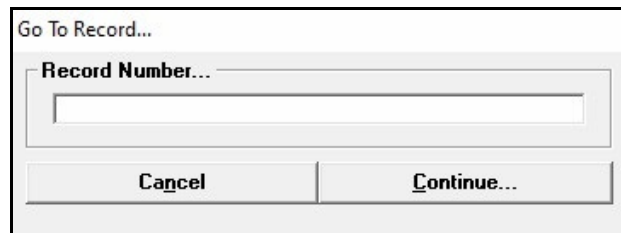
● Some options in the Solution routine require that a single gas properties file be saved for the model. This screen may be used to create the required file - click the *Save* command button and enter the model name for the gas properties file name.

● The gas properties values are calculated at the model's currently set base conditions. To change the base condition values, use the *Other Settings* data tab on the Solution Data screen.

● The combined sum of the mole percent values of all the individual components must equal one hundred (100.00). Use the *Other* data field to enter any remainder required to force a perfect total. The *Other* data field can also accommodate a component unavailable in the gas composition. The *Other* data field has no influence on the calculated values; it is only used to allow for a perfect composition total.

Go To Record

The Go To Record routine is used by many of the report screens. It finds and displays the data associated with a specified Record Number. The Go To Record specification screen is accessed by clicking the *Go To Record* icon from the associated report screen. The features associated with the screen are described as follows.

Data Items

Record Number - Specifies the number of the record to display at the top of the associated report.

Command Buttons

Cancel - Closes the screen without proceeding with the Go To Record routine.

Continue - Proceeds with the Go To Record routine.

Notes & Considerations

● When a value larger than the total record count is entered, the last record will be displayed.

Graphic Settings

The control variables for the GDI Display may be set using the Graphic Settings screen. The screen is accessed by selecting the *Settings* item from the *Graphics* menu or the *Graphic Settings* item from the Data Panel. The features associated with the screen are described as follows.

Data Items

Options -

> **Allow Data Entry During New Feature Entry -** When selected, the associated data may be entered while adding a new feature.

> **Always Display Flags -** When selected, the feature flags will always be displayed. Otherwise, the feature flags will display or be hidden depending on the zoom scale.

> **Automatically Pan & Zoom -** When selected, the pan and zoom functions will be automatically executed by certain mouse events.

Automatically Repaint GDI Image - When selected, the GDI Image will be automatically repainted (redrawn) after certain GDI Commands are executed. When not selected, the GDI Image must be manually repainted using the *Repaint GDI Image* command.

Automatically Update Pipe Length - When selected, the length of a pipe feature will be automatically updated when a command affecting its graphic length is used. When not selected, a prompt will appear after a change that may affect a pipe's length is performed, asking whether or not to update the pipe's length.

Automatically Zoom On Find - When selected, the GDI Display will be automatically zoomed so that the flagged feature's location is centered in the GDI Display when using one of the "*Find*" commands. When not selected, the User must manually zoom to the flagged location.

Display Axis Icon - When selected, the axis icon will be displayed in the lower-left corner of the GDI Display.

Display Flow Arrow Once Per Polyline Pipe - When selected, the flow arrows will only be displayed once per polyline pipe feature. When not selected, the flow arrows will be placed on each segment of a polyline pipe feature.

Display Grips - When selected, the grip symbols are displayed when a feature is highlighted (selected) in the GDI Display. When the grips are displayed, they may be used to move the location of the associated feature by clicking inside the grip symbol and moving it to a new location.

Display Intersection Flags - When selected, the flags resulting from the *Flag Unbroken Intersections* command are displayed in the GDI Display.

Display Reference Grid - When selected, the GDI Reference Grid is displayed. When the reference grid is displayed, the *Snap Grid Intersections* command may force newly entered features to snap to the grid intersection points.

Display Running Pipe Length - When selected, the length of a pipe feature, as it is being added, will be displayed. The length will be displayed in the information panel to the right of the XY coordinate values.

Display Solid Pipe Symbols - When selected, the pipe symbols will be filled (solid) with the current pipe symbols color. When not selected, hollow symbols will be displayed.

Display Vertex Symbols - When selected, a symbol will be placed at each vertex location along a polyline pipe feature or a polyline customer service line feature.

Ignore Pipes Outside GDI Display - When selected, this item may help reduce the time required to repaint the GDI Image.

Show Composite Service Size Change Symbol - When selected, a symbol or marker will be displayed at the location of the pipe size change along a composite-style service line. This option is only available in the UKDM version of the GASWorkS software.

Suppress Flow Arrow Where Flow Is Less Than Convergence Tolerance - When selected, the display of the flow arrows will be suppressed on pipe features where the flow rate value is less than the Convergence Tolerance. When not selected, flow arrows will be placed on all pipe features where the absolute flow value is greater than zero.

Use 3D Coordinates (X,Y,Z) - When selected, prompts for coordinate values will expect X, Y, and Z values to be entered. When not selected, only X and Y values are required.

Vary Resolution On Zoom - When selected, the drawing of features whose display size is very small will be suppressed. Selecting this item may reduce the time required to redraw the GDI Image.

Feature Highlight Color - Specifies the color used to display a highlighted (selected) feature. To set a Color, click in the associated box. The Color Palette will be displayed. Select the desired color, then click the *OK* command button to assign the selected value.

Use Default Highlight Color - When selected, a color similar to the "opposite" of the selected feature's current color will be used to highlight the feature. When unselected, the selected *Feature Highlight Color* will be used.

Settings - See the following *Notes & Considerations* section for comments on the effect of *Settings* items.

Customer Symbol Display Limit - Specifies the display limit for the customer symbols.

Customer Symbol Size - Specifies the default size of the symbol used to display the customer features. The size may be expressed as an absolute value in the *Coordinates* units (select *In Model Coordinate Units*) or relative to the diagonal distance across the GDI Display expressed as a percentage (select *% Of Display Width*).

Customer Symbol Style - Specifies the default style of the symbol used to display the customer features.

Feature Snap Ratio - Specifies the sensitivity of the *Feature Snap* command. The value is expressed as a factor.

Flow Arrow Display Limit - Specifies the display limit for the flow arrow symbols.

Flow Arrow Size - Specifies the symbol size used to display the pipe flow arrows. The size may be expressed as an absolute value in the *Coordinates* units (select *In Model Coordinate Units*) or relative to the diagonal distance across the GDI Display expressed as a percentage (select *% Of Display Width*).

Flow Arrow Style - Specifies the symbol style used to display the pipe flow arrows.

Grip Symbol Size - Specifies the size used to display the grip symbols. The size is expressed as a value relative to the diagonal distance across the GDI Display as a percentage (*% Of Display Width*).

Node Symbol Display Limit - Specifies the display limit for the node symbols.

Node Symbol Size - Specifies the default size of the symbol used to display the nodes. The size may be expressed as an absolute value in the *Coordinates* units (select *In Model Coordinate Units*) or relative to the diagonal distance across the GDI Display expressed as a percentage (select *% Of Display Width*).

Node Symbol Style - Specifies the default style of the symbol used to display the node locations.

Pan/Scroll Change - Horizontal - Specifies the amount to move the GDI Image when panning horizontally (right or left) using the *Scroll* icon. The value is expressed as a percentage of the GDI Display width.

Pan/Scroll Change - Vertical - Specifies the amount to move the GDI Image when panning vertically (up or down) using the *Scroll* icon. The value is expressed as a percentage of the GDI Display width.

Pipe Line Style - Specifies the default line style used to display the pipe features.

Pipe Line Thickness - Specifies the default line thickness (width) used to display the pipe features. The value is expressed in screen pixels.

Pipe Symbol Display Limit - Specifies the display limit for the pipe symbols.

Pipe Symbol Size - Specifies the symbol size used to display the non-pipe hydraulic features (compressor, regulator, valve, and well). The size may be expressed as an absolute value in the *Coordinates* units (select *In Model Coordinate Units*) or relative to the diagonal distance across the GDI Display expressed as a percentage (select *% Of Display Width*). Or, enter zero (0) to indicate that the pipe symbol size will be determined by the distance between the From Node and To Node.

Reference Grid Color - Specifies the color used to display the GDI Reference Grid. To set a Color, click on the associated cell. The Color Palette will be displayed. Select the desired color, then click the *OK* command button to assign the selected value.

Reference Grid Spacing - Specifies the distance between lines in the reference grid.

Refresh Increment - Specifies how often the GDI Display should be updated when the GDI Image is being repainted. The value is expressed as a percentage of the image contents. The smaller the value, the more often the GDI Display will be updated. The larger the value, the less often the GDI Display will be updated. If the value is set at 100, the GDI Display will not be updated until the GDI Image is completely repainted. Adjusting the value may reduce the redraw time.

Supply Flag Size - Specifies the size of the flag symbol used to identify the supply nodes. The size may be expressed as an absolute value in the *Coordinates* units (select *In Model Coordinate Units*) or relative to the diagonal distance across the GDI Display expressed as a percentage (select *% Of Display Width*).

Zoom Ratio - Specifies the sensitivity of the *Zoom In* and *Zoom Out* commands. The value is expressed as a factor.

Command Buttons

Apply - Closes the screen and applies changes.

Cancel - Closes the screen without saving changes.

Save As Default - Saves the current Graphic Settings as the default settings for creating a new model.

Notes & Considerations

● The Feature Snap Ratio is used to adjust the sensitivity of the *Feature Snap* command. The product of the *Feature Snap Ratio* value and the *Node Symbol Size* value determines the size of the "snap target." When adding or changing certain features or measuring a distance, if the Feature Snap is turned "on," the GDI will attempt to find an existing node within the *Snap Target* value of the selected point.

● The Zoom Ratio indicates the magnitude of change in the GDI Image size, which occurs when either *Zoom In* or *Zoom Out* commands are used. For example, if the Zoom Ratio is set at two (2), the *Zoom In* command will increase the size of the GDI Image by two times. Conversely, if the Zoom Ratio is set to two (2), the *Zoom Out* command will decrease the size of the GDI Image by two times.

● Grips are small boxes displayed at various feature reference points when a graphic feature is selected. For example, the small boxes that appear at the pipe ends, vertices, or text insertion points when a pipe is selected. Grips may be used to graphically move data features displayed in the GDI by left-clicking within the box representing the grip, then moving the mouse to the new location and left-clicking the mouse again.

● When the *Vary Resolution On Zoom* graphic setting is selected, the features where the display size is less than one screen pixel will not be displayed. This option will reduce the time required to draw/redraw the GDI Image. When selected, the image may appear to be missing some features when zoomed "out." This occurs because the display of features may be ignored at the particular zoom scale.

● Display Limit values suppress the display of the associated feature, reducing "clutter" in the image. When the diagonal distance across the screen is greater than the display limit, the associated feature *is not* displayed. If the diagonal distance across the screen is less than the display limit, the associated feature *is* displayed.

Header Data Report

The Header Data Report documents the various solution options and other data associated with the model. The Header Data Report is accessed by selecting the *Header Data* item from the *Report* menu. The features associated with the report are described as follows.
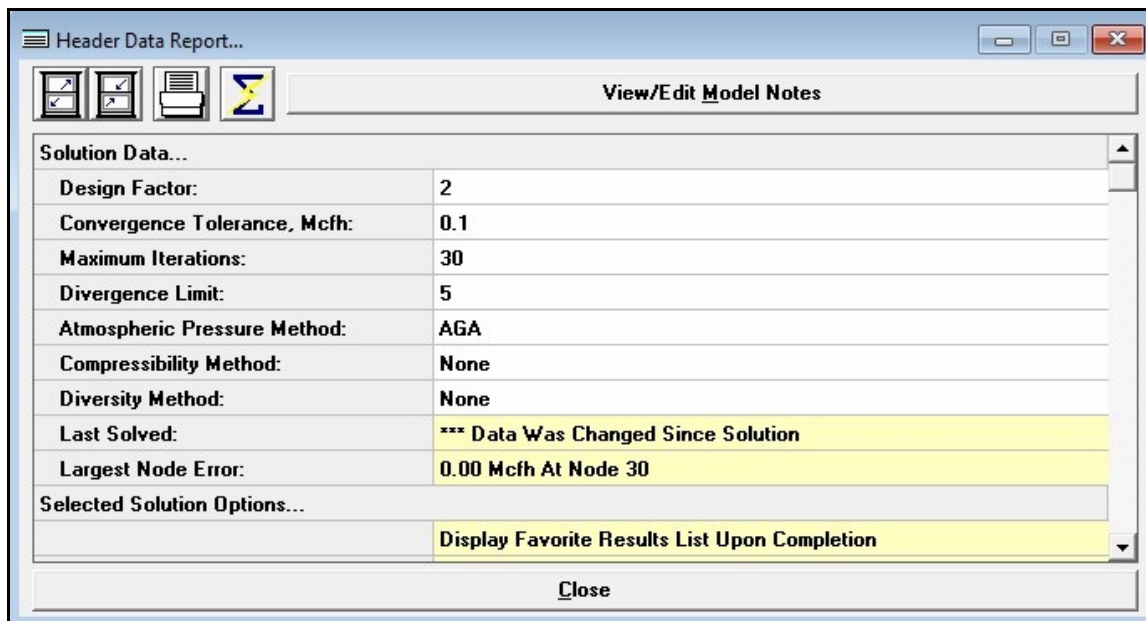
Displayed Items

Header Data Items - Displays the contents of the Header Data Report. To browse the display, adjust the vertical scroll bar adjacent to the display area.

Icons
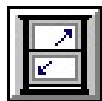
Maximize Screen Size - Increases the Header Data Report's size to fill the GASWorkS display.

Print - Prints the contents of the Header Data Report. When clicked, the Print Settings screen is displayed.

Quick Solve - Solves the model using the currently set solution options.

B3PE LLC *Engineering & Software Services*
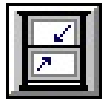
Reset Screen Size - Sets the Header Data Report's size to its "normal" dimensions and position.

Command Buttons

Close - Closes the Header Data Report.

View/Edit Model Notes - Displays the Model Notes screen.

Notes & Considerations

● In the report, items displayed with a "white" background may be entered or changed directly from the Header Data Report. Items with a "colored" background cannot be changed.

● If the GDI Window is open, the node with the largest flow error may be flagged by double-clicking on either the *Largest Node Error* label or the associated cell. A flag will be placed at the node's location, and the display will be "zoomed" to that location.

Mass Update Specifications

Mass changes may be made to the model data using the Mass Update routine. This type of change is referred to as a "mass update." The Mass Update routine is controlled using the Mass Update Specifications screen. The screen is accessed by selecting the *Mass Update* item from the *Edit* menu. The features associated with the screen are described as follows.

Data Items

Apply To All Records - Specifies that the Mass Update will be applied to all associated data records.

Apply Only To A Selection Set - Specifies that the Mass Update will only be applied to a User-specified set of records.

Equal To - Specifies the new value for the specified *Set* item. The characteristics and contents of this item will change depending on the selected *Set* item.

Is - Specifies the search operator to be used during the Mass Update. The search operator specifies the relation between the *Where* and *To* values.

Item Type - Specifies the type of data feature to be changed.

Saved Specifications - Displays previously saved mass update specifications. When a specification is selected, the values on the screen will be replaced with the values associated with the selected specification.

Set - Specifies the data item to be updated.

To - Specifies the check value used during the matching record search. The characteristics and contents of this item will change depending on the selected *Where* item.

Where - Specifies the data item to be updated.

Command Buttons

Close - Closes the screen.

Delete Displayed Specification - Deletes the currently selected specification in the *Saved Specifications* list.

Restore - Restores the data values that existed just before the executed update. The *Restore* command button will only be enabled when the *Allow Undo Of Data/Graphic Changes* preference settings option is selected.

Save Current Specification - Saves the currently displayed update specifications.

Update - Performs the Mass Update using the User-specified settings and data values.

Notes & Considerations

● If the *Allow Undo Of Data/Graphic Changes* preference settings option is selected, changes to the model data may be "undone" by immediately clicking the *Restore* command button after the Mass Update was performed.

● See the *Query Features* topic in the *Miscellaneous* section for the meanings of the search operators.

Model Notes

Notes and reference information about an analysis or model may be kept using the Model Notes feature. The Model Notes may be accessed by either selecting the *Model Notes* item from the *Edit* menu, by clicking the *View/Edit Model Notes* command button from the Header Data Report, or the *Model Notes* data tab on the Solution Data screen. The features associated with the screen are described as follows.
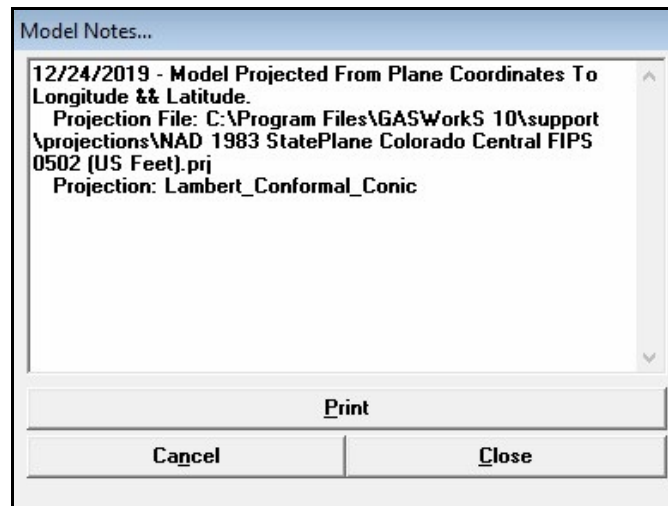
Data Items

Model Notes - Used to display and enter the Model Notes associated with the currently open model.

Command Buttons

Cancel - Closes the screen without saving changes.

Close - Closes the screen and saves changes.

Print - Prints the contents of the notes. When clicked, the Print Settings screen is displayed.

GASWorkS™ 10.1

Data Screens

Notes & Considerations

● The Model Notes editor supports standard Windows cut-and-paste commands (*Ctrl-C*, *Ctrl-X*, and *Ctrl-V*) and the other standard editing commands controlled by the *Enter*, *Backspace*, *Delete*, and *Arrow* keys. The editor will automatically wrap the text if the line is longer than the available viewing space. To cause a specific line break, press the *Enter* key at the end of the desired line. The *Tab* keys do not function within the editor.

Model Selection

The Model Selection screen is used to access the model data files. The Header file (.hdr) associated with a model is the "key" file for accessing the model data. It is the only file type listed when using the Model Selection screen. The Model Selection screen is displayed by selecting the various items in the *File* menu. The features associated with the screen are described as follows.

Displayed Items

Model Preview - Displays a preview of the selected model.

Model Statistics - Displays the various information about the selected (highlighted) model. The contents of the display will vary depending on the version of the selected model.

Data Items

Directories - Specifies the directory (folder) on the selected disk drive. When a directory has been selected, any models in that directory will be displayed in the *Files* list.

Drives - Specifies the disk drive to use. When a drive is selected, the directories contained on that drive will be displayed in the *Directories* list.

Model Name - Specifies the name of the model to be used. A list of the most previously used models may be accessed from the drop-down list. The Model Name may be entered by various means. If the Model Name is manually entered, include the path.

Models - Specifies the models found in the selected directory on the selected disk drive.

Command Buttons

Cancel - Closes the screen without saving changes and without proceeding with the routine that invoked the screen.

Continue - Accepts the displayed model name and proceeds with the associated routine.

Notes & Considerations

● Depending on the routine that invoked the Model Selection screen, the response to errors that may occur while selecting a model will vary. If the routine requires a new model name and an existing one is selected, the screen will prompt whether the model should be overwritten - respond appropriately. If an existing model is chosen to be "overwritten," the model is not destroyed; it is sent to the Windows Recycle Bin. If the routine requires the name of an existing model and the name of a non-existent model is specified, the screen will respond with either a message that the model does not exist.

● Wild card characters may be entered in the *Model Name* data field to alter the contents of the *Models* panel. Valid characters are "*" for multi-character replacement and "?" for single-character replacement.

● *Drives*, *Directories*, and *Model Name* values must correspond to the standard "DOS" format. That is, drive letters must be followed by a colon ":" directories must be separated by a backslash "\," and extensions must be proceeded by a dot ".".
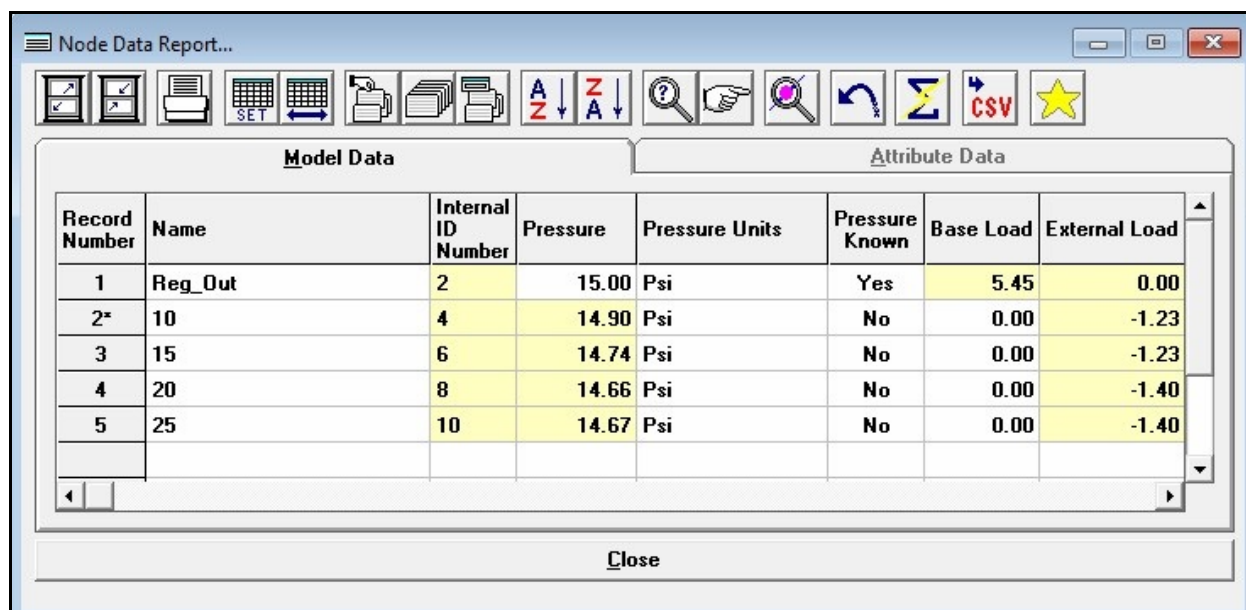
● When manually entering a value in the *Model Name* data field, include the drive, full path name, and model name.

● The name of the currently open model is displayed at the top of the main GASWorkS screen.

● To save a model to a directory (folder) other than the "default" directory (folder), select the desired location using the drive and directories, then enter the model name only in the model name data field.

Node Data Report

The Node Data Report is accessed by selecting the *Node Data* item from the *Report* menu. The "report" displays both entered and calculated data values. Entered data values may be changed directly in the report by the User. The features associated with the report are described as follows.

Model Data Tab

Data Items

Adjust Base Load By DF - Indicates whether the node's base load value will be multiplied by the Design Factor during the Solution routine. The original data value is not changed by this setting; the *Base Load* value is only modified during the Solution routine.

Adjust External **Load By DF -** Indicates whether the node's external load value will be multiplied by the Design Factor during the Solution routine. The original data value is not changed by this setting; the *External Load* value is only modified during the Solution routine.

Base Load - Specifies the node's base load value. If the *Base Load* value is "known" (*Base Load Known* set to "Yes"), enter a value. If the *Base Load* value is "unknown," set the *Base Load Known* item to "No." If the model has been recently solved and the load is "unknown," the item will display the calculated load value for the node. Initially, the field is populated with the default *Node Base Load* value. See the following *Notes & Considerations* section for additional information on the *Base* and *External Load* values.

Base Load Known - Indicates whether the base load value is "known" ("Yes") or "unknown" ("No").

Base Load Status - Indicates whether the base load value is "on" or "off" during the Solution routine. If the status is "off," the *Base Load* value is assumed to be zero (0) during the Solution routine. The original data value is not changed by this setting.

Calibration Type - Specifies the type of calibration style.

Calibration Value - Specifies the known value to be checked against. The value is expressed in the *Pressure Units* or *Load Units*, depending on the selected *Calibration Type*.

Elevation - Specifies the node's height above sea level. Initially, the field is populated with the default *Node Elevation* value.

External Load Status - Indicates whether the external load value is "on" or "off" during the Solution routine. If the status is "off," the *External Load* value is assumed to be zero (0) during the Solution routine. The original data value is not changed by this setting.

Gas Properties **Known -** Indicates whether the gas properties are "known" ("Yes") or "unknown" ("No").

Heating Value - Specifies the heating value of the gas at the node's location.

Load Units - Specifies the dimensional units for the various load values.

Name - Specifies the unique name or number for the associated node. The name may contain up to forty (40) alphanumeric characters. The name is automatically assigned when a new node is added. However, the User can change the name to the desired value. The name must be unique to the model. A message will be displayed if the name already exists.

Pressure - Specifies the node's pressure value. If the *Pressure* value is "known" (*Pressure Known* set to "Yes"), enter a value. If the *Pressure* value is "unknown," set the *Pressure Known* item to "No." Initially, the field is populated with the default *Node Pressure* value. If the model has been recently solved and the pressure is "unknown," the item will contain the calculated *Pressure* value for the node.

Pressure Known - Indicates whether the pressure value is "known" ("Yes") or "unknown" ("No").

Pressure Units - Specifies the dimensional units for the *Pressure* value.

Specific Gravity - Specifies the specific gravity value of the gas at the node's location. The Specific Gravity represents the weight or density of the gas compared to dry air.

Specific Heat - Specifies the specific heat ratio of the gas at the node's location. The specific heat ratio represents the constant pressure specific heat divided by the constant volume specific heat of the gas mixture.

Temperature - Specifies the average flowing temperature of the gas at the node's location. If the *Temperature* value is "known" (*Temperature Known* set to "Yes"), enter a value. If the *Temperature* value is "unknown," set the *Temperature Known* item to "No." Initially, the field is populated with the default *Node Temperature* value. If the model has been recently solved, the *Temperature* value is unknown, and the *Calculate Temperature Values* solution option was selected, the item will contain the calculated *Temperature* value for the node.

Temperature **Known -** Indicates whether the temperature value is "known" ("Yes") or "unknown" ("No").

Viscosity - Specifies the dynamic viscosity of the gas as measured (calculated) at the node's location.

X-Coordinate - Specifies the node's east-west coordinate value (horizontal location in plan view).

Y-Coordinate - Specifies the node's north-south coordinate value (vertical location in plan view).

Z-Coordinate - Specifies the node's vertical coordinate value when using 3D Coordinates.

Calculated Values

Atmospheric **Pressure -** Displays the calculated atmospheric pressure value based on the node's elevation.

Calibration Difference - Displays the difference between the known value and the value in the model.

Calibration Difference, % - Displays the difference, as a percentage, between the known value and the value in the model.

Edit Date - Displays the date when the node data was last changed.

External Load - Displays the external load value times the Design Factor if the *Adjust Ext Load By DF* item is "Yes."

External Load (Fixed) - Displays the portion of the external load that cannot be adjusted by the Design Factor.

Flow Error - Displays the difference between the total flow into the node and the total flow out of the node. See the following *Notes & Considerations* section for additional information.

Internal ID Number - Displays the node feature's unique (to GASWorkS) identification number. This value is automatically assigned and cannot be changed by the User.

Total Load - Displays the total of the (adjusted) *Base*, *External Load*, and *External Load (Fixed)* values.

Attribute Data Tab

If a node attribute file is attached to the model, the *Attribute Data* tab of the Node Data Report displays the data contained in the attribute file. If a database is attached, this data tab will display the values associated with the data fields contained in the associated attribute file (database) and will vary by file. If a "linked" style database is attached, the values may be changed by the User. If a "non-linked" style database is attached, the values are view-only and cannot be changed by the User. The displayed items (columns) will vary depending on the content of the attached file.

Displayed Items

Record Number - Refers to the position of the current node record in the node data file. An asterisk ("*") next to the Record Number denotes that changes have been made to the record since the last execution of the Solve routine. This value is automatically assigned and cannot be changed by the User.

Data Items

Link ID Number - Specifies the identification number used to link the node records in the model to the records in an attached attribute file. When a "linked" style attribute file is attached, the number is automatically assigned and cannot be changed by the User. When a "non-linked" style attribute file is attached, the number is the same as the node name value assigned by the User. The number should represent a unique value in the attribute file (database).

Link To Database - Indicates whether the node should be linked to the attribute file (database). If "Yes" is selected, the node will be linked to the database by matching the Link ID Number.

Icons

Add To Favorites - Adds the selected (highlighted) node record to the *Favorite Results* list.

Edit Selection Set Values - Changes data values en masse for the current selection set of nodes. When clicked, the Multiple Edit Specifications screen is displayed.

Find Node - Finds and flags the selected (highlighted) node record in the GDI Display. This icon only works if the GDI Window is open before the Node Data Report is opened.

Find Record - Finds a node record in the current selection set by node Name. When clicked, the Find Record specification screen is displayed.

Go To Record - Displays the node record associated with a specified Record Number at the top of the current data tab. When clicked, the Go To Record specification screen is displayed.

Make Selection Set - Creates a selection set of nodes that meet User-specified selection criteria. When clicked, the Query Specifications screen is displayed.
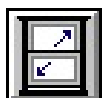
Maximize Screen Size - Increases the Node Data Report's size to fill the GASWorkS display.

Print - Prints the contents of the displayed data tab. When clicked, the Print Settings screen is displayed.

Quick Solve - Solves the model using the currently set solution options.
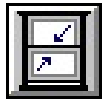
Reset Screen Size - Sets the Node Data Report's size to its "normal" dimensions and position.

Reset Selection Set - Resets (clears) the current selection set. When clicked, all reportable node records will be displayed on the associated data tab.
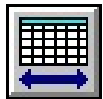
Restore Columns - Restores the display of all columns (data items). When clicked, all of the columns associated with the current data tab will be displayed.

Save To CSV File - Saves the data associated with the current data tab to a comma-separated values file (.csv). When clicked, the file will be created, and a message will be displayed stating the location of the saved file.

Set Columns - Used to specify which columns (data items) are displayed in the Node Data Report. When clicked, the Columns specification screen is displayed.

Sort Ascending - Sorts the records in ascending order (A to Z or lowest to highest value) based on the node Name.

Sort Descending - Sorts the records in descending order (Z to A or highest to lowest value) based on the node Name.

Undo - Restores the most recent data change.

Command Buttons

Close - Closes the Node Data Report.

Notes & Considerations

● The Flow Error values displayed in the report represent the difference between the total flows into and out of a node. For a "solved" or "balanced" model, the magnitude of these values will be no greater than the *Convergence Tolerance* value in the Solution Data. For an "unsolved" or "unbalanced" model, these values may be used to determine where the solution is having issues trying to solve. Larger Flow Error values are associated with nodes where the model has trouble solving. Check the model values and connectivity near these locations for errors.

● The *Favorites List* in the GDI Data Panel may be used to display select values for select nodes, allowing quick access to results upon completion of the solution. The Favorites List can provide an alternative to browsing the report for frequently viewed node records.

● The dimensional unit for the atmospheric pressure is automatically set to the same dimensional unit as the base pressure dimensional unit (set on the Solution Data screen).

● The displayed base load represents the unadjusted base load. The displayed external load represents the adjusted value. The displayed total load represents the sum of the adjusted base, adjusted external, and fixed *External Load* values.

● Normally, the report will contain more information than will fit on a single page. If this condition occurs when trying to print, a message will be displayed. When the message is displayed, respond appropriately. To print a report that is too wide for the printer, either eliminate some of the undesired columns using the Set Columns routine or by manually collapsing the undesired columns, set the printer to use *Landscape Orientation*, or reduce the printer font size.

● The *Report* item in the Facility Settings controls which features are included in the Node Data Report. Only nodes attached to pipes assigned to Facility Types that are allowed to be reported are included in the Node Data Report. The Facility Settings may be viewed and changed using the *Facility Settings* item found in the *Edit* menu or the *Report Options* item from the *Report* menu.

● When a ".csv" file is created, it contains the data and calculated values for records in the current selection set and only for the currently displayed columns. The file may be opened by any application which supports the ".csv" file format.

● A cell with a "grey" background contains a fixed value. A cell with a "white" background contains a User-entered or selected value. A cell with a "yellow" background contains a calculated value. A cell with a "cyan" background represents an unused or not applicable item. Only values in cells with a "white" background may be changed by the User.

Path Settings

The default directory (folder) locations for the various files associated with a GASWorkS model and the support files are controlled using the Path Settings screen. The Path Settings screen is accessed by selecting the *Paths* item from the *Preferences* submenu of the *File* menu. The features associated with the screen are described as follows.
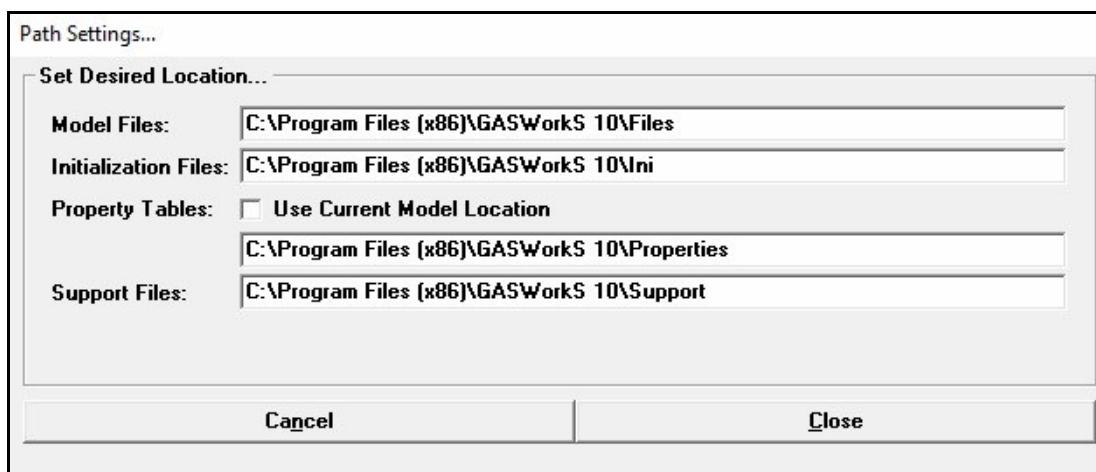
Data Items

Initialization Files - Specifies the default location for the associated initialization files.

Model Files - Specifies the default location for the associated model files.

Property Tables - Specifies the default location for the associated Property Tables.

Support Files - Specifies the default location for the associated GASWorkS support files.

Use Current Model Location - When selected, the current model location will be used for the Property Tables.

Command Buttons

Cancel - Closes the screen without saving changes.

Close - Closes the screen and saves changes.

Notes & Considerations

● To change a path setting, double-click in the data field to display the Path Selection screen. Use the *Drive* and *Directories* lists to change the current setting, then click the *Continue* command button to save the selection.

● The Model Files path may be "overridden" when selecting a file using the Model Selection screen. All other paths indicate locations GASWorkS uses to find the necessary support files. These directories can only be changed using the Path Settings screen.

● The Initialization Files contain information about the screen location and previous data values and settings for each screen.

● The Property Tables contain dimensional data and other property data for the various pipes and devices used in the GASWorkS calculations.

● The Support Files contain information used by various routines, including:
- • Block and template definitions used by the DXF Export routine
- • Coordinate projection files
- • Default and example data lists
- • Default and example database definitions
- • Graphic images
- • Default and example settings and model templates

● The Path Settings may be saved to a "default" file using the *Save Current Settings* item from the *Preferences* submenu of the *File* menu. Once saved, the file may be opened and used during other GASWorkS sessions. This is useful if a particular system or client requires a certain group of setting values; the settings may be saved and opened when needed in the future.

● Universal Naming Convention (UNC) names may be manually entered for any path location. Ensure the name is valid and entered correctly - for example, "\\network_drive\c\model_files."

Pipe Data Report

The Pipe Data Report is accessed by selecting the *Pipe Data* item from the *Report* menu. The "report" displays both entered and calculated data values. Entered data values may be changed directly in the report by the User. The features associated with the report are described as follows.

Model Data Tab

Displayed Items

Edit Date - Displays the date when the pipe data was last changed.

Internal ID Number - Displays the pipe features unique (to GASWorkS) identification number. This value is automatically assigned and cannot be changed by the User.

Data Items

Allow Sizing - Indicates whether the pipe size may be calculated during the Solution routine when the *Calculate Pipe Sizes (Optimize)* solution option is selected. This item is only enabled for *Pipe* Hydraulic Types.

Attached Fittings - Indicates the number of fittings that are currently attached to the associated record. Click in the associated cell to display the Fitting Data screen. Make the required changes, then click the *Close* command button to attach the fittings. This item is only enabled for *Pipe* Hydraulic Types.

Control Style - Specifies whether the set pressure for a compressor-type feature is to be applied to the downstream (discharge) node or the upstream (suction) node. This item is only enabled for *Compressor* Hydraulic Types.

D/R Units (Diameter/Roughness) **-** Specifies the diameter and roughness dimensional units for the associated pipe-type features. This item is only enabled for *Pipe* Hydraulic Types.

Efficiency - Specifies the efficiency value. Initially, the efficiency is set to the default *Pipe Efficiency* value. For pipe and fitting-type features, this item refers to the Hydraulic Efficiency of the feature. For compressor-type features, this item refers to the Mechanical Efficiency of the compressor. This item is only enabled for *Compressor*, *Fitting*, and *Pipe* Hydraulic Types.

Facility Type - Specifies the Facility Type of the associated record.

Flow Equation - Specifies the flow equation for calculating pressure and flow values for the associated record.

Hydraulic Type - Specifies the hydraulic element type for the record.

Length - Specifies the hydraulic length for the associated record. Initially, the value is set to the pipe's "graphic" length. This item is only enabled for *Pipe* Hydraulic Types.

Length Units - Specifies the length dimensional units for the associated pipe-type feature. This item is only enabled for *Pipe* Hydraulic Types.

Number Of Wells - Specifies the number of wells represented by the associated record. This item is only enabled for *Well* Hydraulic Types.

Pressure Units - Specifies the dimensional units of the pressure values.

Q Units (Flow) **-** Specifies the dimensional units of the flow rate values.

Set Pressure - Specifies the set pressure value for a compressor or regulator-type feature. This item is only enabled for *Compressor* and *Regulator* Hydraulic Types.

Set Pressure Units - Specifies the dimensional units for the *Set Pressure* value. This item is only enabled for *Compressor* and *Regulator* Hydraulic Types.

GASWorkS™ 10.1 *Data Screens*

Table Of Contents

Size/Type Code - Specifies the Size/Type Code. The meaning of this item will vary depending on the selected *Hydraulic Type*. The various meanings are listed in the following table:

Hydraulic Type	Size/Type Definition
Compressor	Represents a code for the compressor size and type. The content of the list represents the Size/Type Codes found in the currently selected Compressor Property Table.
Fitting	Represents a code for the fitting size and type. The content of the list represents the Size/Type Codes found in the currently selected Fitting Property Table.
Pipe	Represents either the Size/Type Code or the *Inside Diameter* value for a pipe-type feature. Select the desired Size/Type Code, or enter the *Inside Diameter* value. The content of the list represents the Size/Type Codes found in the currently selected Pipe Property Table.
Regulator	Represents a code for the regulator size and type. The content of the list represents the Size/Type Codes found in the currently selected Regulator Property Table.
Valve	Represents a code for the valve size and type. The content of the list represents the Size/Type Codes found in the currently assigned Valve Property Table.
Well	Represents a code for a well record. The content of the list represents the Size/Type Codes found in the currently assigned Well Property Table. If the "Unique" item is selected, the required well parameters must be manually entered in the appropriate data fields.

Sizing Group - Specifies the Sizing Group to associate the current pipe-type feature. When using the Pipe Sizing routine, only pipe sizes associated with the assigned group will be considered during the pipe selection. This item is only enabled for *Pipe* Hydraulic Types when the *Allow Sizing* item is "Yes."

Status - Indicates whether the *Hydraulic Type* is turned "on" or "off."

Valve Opening - Specifies the opening position for a valve-type feature. The value is expressed as a percentage, with 0 (zero) being completely closed and 100 being completely open. This item is only enabled for *Valve* Hydraulic Types.

Well Equation Coefficient - Specifies the equation coefficient for a well-type feature. This item is only enabled for *Well* Hydraulic Types when the *Size/Type Code* is "Unique."

Well Equation Exponent - Specifies the equation exponent for a well-type feature. This item is only enabled for *Well* Hydraulic Types when the *Size/Type Code* is "Unique."

Calculated Values

% (Percent) **SMYS -** Displays the ratio of the *Hoop Stress* value to the *Specified Minimum Yield Strength* (SMYS) value for pipe-type features. The value is based on the SMYS value found in the Pipe Property Table. The value is not displayed if the SMYS value is zero (0) or the Size/Type Code is not found in the Pipe Property Table. This value is only displayed for *Pipe* Hydraulic Types.

Average Compressibility - Displays the average compressibility factor. The value is calculated using the *Average Pressure* value and the Compressibility calculation method selected in the Solution Data.

Average Pressure - Displays the average pressure for the hydraulic type.

Customer Count - Displays the number of customers attached to the associated pipe feature.

Drop Per XXX - Displays the unit pressure drop along the pipe segment for pipe-type features. This value is calculated by dividing the overall pressure drop across the segment by the segment's hydraulic length and then multiplying the result by the specified *Pressure Drop Per Length*. The *Pressure Drop Per Length* value may be set on the *Other* data tab on the Report Options screen. This value is only displayed for *Pipe* Hydraulic Types.

Equivalent Length - Displays the equivalent length value for attached fittings on pipe-type features. This value is only displayed for *Pipe* Hydraulic Types that have a fitting attached.

Flow Rate - Displays the calculated flow rate value. A positive value indicates that the flow direction is from the From Node to the To Node, and a negative value indicates that the flow direction is from the To Node to the From Node.

Hoop Stress - Displays the calculated hoop stress value for pipe-type features. The value is based on the larger From Node or To Node *Pressure* values and the *Outside Diameter* and *Wall Thickness* values found in the Pipe Property Table. This item is not displayed if the Size/Type Code is not in the Pipe Property Table. This value is only displayed for *Pipe* Hydraulic Types.

Inlet Pressure - Displays the inlet pressure value for the associated pipe features.

Inside Diameter - Displays the inside diameter value for pipe-type features. If the Size/Type Code represents a code, this value represents the *Inside Diameter* value extracted from the assigned Pipe Property Table. Otherwise, this item represents the interpreted numeric Size/Type Code value. This value is only displayed for *Pipe* Hydraulic Types.

Outlet Pressure - Displays the outlet pressure value for the associated record.

Pressure Drop - Displays the linear pressure drop across a pipe segment or device (the From Node Pressure minus the To Node Pressure). *Note* - When elevation differences are present along the pipe segment, it is possible for the downstream pressure to be larger than the upstream pressure. A positive value indicates that the From Node pressure is larger than the To Node pressure, and a negative value indicates that the To Node pressure is greater than the From Node pressure.

Reynolds Number - Displays the Reynolds Number for pipe-type features. This value is only displayed for *Pipe* Hydraulic Types.

Roughness - Displays the inside pipe wall roughness value for pipe-type features. The value is either extracted from the Pipe Property Table for the assigned Size/Type Code or entered by the User. This value is only displayed for *Pipe* Hydraulic Types.

SMYS - Displays the specified minimum yield strength (SMYS) value for pipe-type features. This value is only displayed for *Pipe* Hydraulic Types.

Total Efficiency **-** Displays the total hydraulic efficiency value for pipe-type features. This item is only displayed for *Pipe* Hydraulic Types.

Velocity - Displays the maximum velocity of the gas flow in pipe-type features. The value is calculated using the lower of the segment's end pressures, the volumetric flow rate, and the *Inside Diameter* value for the pipe feature. This value is only displayed for *Pipe* Hydraulic Types.

Volume - Displays the volume of gas contained in pipe-type features. The value is calculated using the *Average Pressure* value, the inside diameter, and the hydraulic length of the pipe segment. This value is only displayed for *Pipe* Hydraulic Types.

Attribute Data Tab

If a pipe attribute file is attached to the model, the *Attribute Data* tab of the Pipe Data Report displays the data in the attribute file. If a database is attached, this data tab will display the values associated with the data fields contained in the associated attribute file (database) and will vary by file. If a "linked" style database is attached, the values may be changed by the User. If a "non-linked" style database is attached, the values are view-only and cannot be changed by the User. The displayed items (columns) will vary depending on the content of the attached file.

Displayed Items

From Node - Refers to the name of the beginning node of the associated pipe feature.

Internal ID Number - Displays the pipe features unique (to GASWorkS) identification number. This value is automatically assigned and cannot be changed by the User.

Record Number - Refers to the position of the current pipe record in the pipe data file. An asterisk ("*") next to the Record Number denotes that changes have been made to the record since the last execution of the Solve routine. This value is automatically assigned and cannot be changed by the User.

To Node - Refers to the name of the ending node of the associated pipe feature.

Data Items

Link To Database - Indicates whether the pipe should be linked to the attribute file (database). If "Yes" is selected, the pipe will be linked to the database by matching the Link ID Number.

Link ID Number - Displays the identification number used to link the pipe records in the model to the records in an attached attribute file or as an identifier for the pipe when no attribute file is attached. When a "linked" style attribute file is attached, the number is automatically assigned and cannot be changed by the User. When a "non-linked" style attribute file is attached, the number is assigned by the User. In this case, the number should generally represent a unique value in the attribute file (database). When no attribute file is attached, the number may be used to maintain any User desired alphanumeric value. The value may be a combination of up to twenty (20) alphanumeric characters.

Icons

Add To Favorites - Adds the selected (highlighted) pipe record to the *Favorite Results* list.

Delete Current Selection - Deletes the current selection set of pipe features.

Delete Pipe - Deletes the selected (highlighted) pipe record.

Edit Selection Set Values - Changes data values en masse for the current selection set of pipe features. When clicked, the Multiple Edit Specifications screen is displayed.

Find Pipe - Finds and flags the selected (highlighted) pipe record in the GDI Display. This icon only works if the GDI Window is open before the Pipe Data Report is opened.

Find Record - Finds a pipe record in the current selection set by either the From Node or To Node Name. When clicked, the Find Record specification screen is displayed.

Go To Record - Displays the pipe record associated with a specific Record Number at the top of the current data tab. When clicked, the Go To Record specification screen is displayed.

Make Selection Set - Creates a selection set of pipes that meet User-specified selection criteria. When clicked, the Query Specifications screen is displayed.
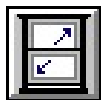
Maximize Screen Size - Increases the Pipe Data Report's size to fill the GASWorkS display.

Print - Prints the contents of the displayed data tab. When clicked, the Print Settings screen is displayed.
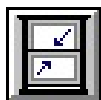
Reset Screen Size - Sets the Pipe Data Report's size to its "normal" dimensions and position.

Reset Selection Set - Resets (clears) the current selection set. When clicked, all of the reportable pipe records will be displayed on the associated data tab.
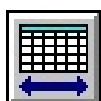
Restore Columns - Restores the display of all columns (data items). When clicked, all of the columns associated with the current data tab will be displayed.

Save To CSV File - Saves the data associated with the current data tab to a comma-separated values file (.csv). When clicked, the file will be created, and a message will be displayed stating the location of the saved file.

Save Current Selection - Saves the current selection set to a new model. The saved pipe records will be used to create a new model.

Set Columns - Used to specify which columns (data items) are displayed in the Pipe Data Report. When clicked, the Columns specification screen is displayed.

Sort Ascending - Sorts the records in ascending order (A to Z or lowest to highest value) based on the From Node.

Sort Descending - Sorts the records in descending order (Z to A or highest to lowest value) based on the From Node.

Undo - Restores the most recent data change.

Command Buttons

Close - Closes the Pipe Data Report.

Notes & Considerations

● Normally, the report will contain more information than will fit on a single page. If this condition occurs when trying to print, a message will be displayed. When the message is displayed, respond appropriately. To print a report that is too wide for the printer, either eliminate some of the undesired columns using the Set Columns routine or by manually collapsing the undesired columns, set the printer to use *Landscape Orientation*, or reduce the printer font size.

● The *Report* item in the Facility Settings controls which data features (records) are included in the Pipe Data Report. Only pipes assigned to *Facility Types* that are allowed to be reported are included in the Pipe Data Report. The Facility Settings may be viewed and changed using the *Facility Settings* item found in the *Edit* menu or the *Report Options* item from the *Report* menu.

● If an attribute file is attached, the contents of the file will be displayed on the *Attribute* tab of the Pipe Data Report. Depending on the type of file attached, the attribute data values may or may not be allowed to be edited.

● When a ".csv" file is created, it contains the data and calculated values for records in the current selection set and only for the currently displayed columns. The file may be opened by any application which supports the ".csv" file format.

● The *Favorites List* in the GDI Data Panel may be used to display select values for select nodes, allowing quick access to results upon completion of the solution. The Favorites List can provide an alternative to browsing the report for frequently viewed node records.

● A cell with a "grey" background contains a fixed value. A cell with a "white" background contains a User-entered or selected value. A cell with a "yellow" background contains a calculated value. A cell with a "cyan" background represents an unused or not applicable item. Only values in cells with a "white" background may be changed by the User.

GASWorkS™ 10.1 *Data Screens*

Preference Settings

Various GASWorkS program behaviors may be set using the Preference Settings screen. The Preference Settings screen is accessed by selecting the *Settings* item from the *Preferences* submenu of the *File* menu. The features associated with the screen are described as follows.
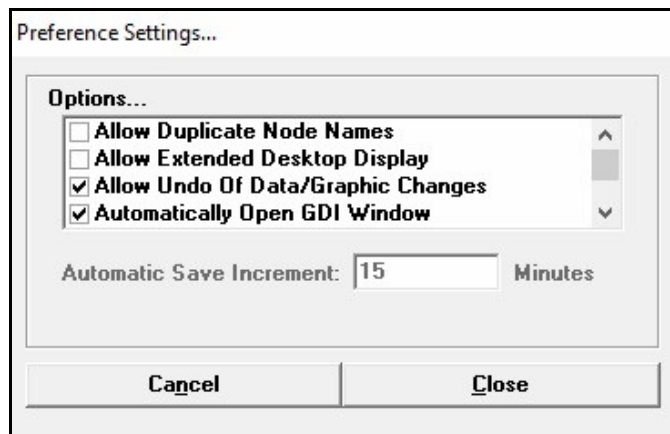
Data Items

Automatic Save Increment - Specifies the time increment between "automatic saves." This item is only enabled when the *Automatically Save At Timed Increments* preference settings option is selected.

Options -

> **Allow Duplicate Node Names -** When selected, duplicate node Names may be used. When not selected, entry of a duplicate (already existing) Node Name is not allowed and will generate an error message.

> **Allow Extended Desktop Display -** When selected, GASWorkS will support extended desktop displays (i.e., dual monitors). Unselect this option if issues occur from a dual monitor setup to a single monitor setup.

> **Allow Undo Of Data/Graphic Changes -** When selected, graphic and data changes may be reversed (undone) using either the *Undo Last Change* command, the *Undo* icon, or the *Restore* command button on the Mass Update Specifications screen. When not selected, changes must be manually restored.

> **Automatically Open GDI Window -** When selected, the GDI Window will automatically be opened each time a model is opened or GASWorkS is opened. When not selected, the GDI window must be manually opened using the *View/Edit* item from the *Graphics* menu.

Automatically Save At Timed Increments - When selected, the model data in memory will be saved to the associated disk files at User-specified time increments. Enter a time value in Minutes in the *Automatic Save Increment* data field. When not selected, the model data must be manually saved.

Display Reminders - When selected, additional messages will be displayed during certain commands and routines to provide reminders to the User. When not selected, the display of most reminder messages will be suppressed.

Display Tooltips - When selected, the tooltips will be displayed when the cursor is rested (hovered) over an icon, menu item, command button, data field, drop-down list, etc. When not selected, the display of tooltip messages will be suppressed.

Ignore Facility Settings On Mass Update - When selected, all data features will be included in the Mass Update routine. When not selected, only items associated with Facility Types set to "Active" will be included in the Mass Update routine.

Save Model Before Solving - When selected, the current model data will be saved to the associated disk files before the Solution routine is executed. When not selected, the model data must be manually saved.

Use Command Alias File - When selected, command alias values will be read from the "Command Alias File" and can be typed on the GDI Command Line. See the *Command Alias File* topic in the *Miscellaneous* section for more information.

Use Last Record For Default Values - When selected, the data values associated with the last record of the same feature type will be used as the default data for a newly added feature type. When not selected, the data values in the *Default Data Values* will be used as the default data for a newly added feature.

Command Buttons

Cancel - Closes the screen without saving changes.

Close - Closes the screen and saves changes.

Notes & Considerations

● When the *Automatically Save At Timed Increments* preference settings option is selected, the User will be prompted to save the model at time increments as specified by the value of the *Automatic Save Increment* data field.

● The Preference Settings may be saved using the *Save Current Settings* item from the *Preferences* submenu of the *File* menu.

Print Settings - Long Form

GASWorkS uses two Print Settings screens - a "long" form and a "short" form. The "Long Form" is displayed when the *Print* icon is clicked from one of the "grid" style report screens. The Print Settings screen allows various printer settings to be set by the User. The features associated with the screen are described as follows.

Displayed Items

Name - Displays the name of the font to be used when printing. Double-click in the data field to open the Font settings screen. Select the desired font, then click the *OK* command button.

Size - Displays the size of the font to be used when printing. Double-click in the data field to open the Font settings screen. Select the desired size, then click the *OK* command button. The size is expressed in printer points. One printer point is approximately equal to 1/72 of an inch.

Status Bar - Displays the status of the Print routine.

GASWorkS™ 10.1   *Data Screens*

Data Items

Bottom Margin - Specifies the distance from the bottom of the paper page to the bottom of the printed page.

Collate Pages - When selected, the pages are collated when multiple copies are printed.

Landscape Orientation - When selected, landscape orientation will be used when printing. When unselected, the pages will be printed using portrait orientation.

Left Margin - Specifies the distance from the left side of the paper page to the left side of the printed page.

Number Of Copies - Specifies the number of copies to be printed. Enter a value greater than zero (0).

Print Grid Lines - When selected, grid lines will be printed between the rows and columns. Printing the row and column grid lines enhances the readability of the report, but their inclusion also increases the print time.

Print Page Numbers - When selected, the page numbers will be printed.

Printer - Specifies the print device to use.

Right Margin - Specifies the distance from the right side of the paper page to the right side of the printed page.

Top Margin - Specifies the distance from the top of the paper page to the top of the printed page.

Command Buttons

Cancel - Closes the screen without printing.

Print - Proceeds with the Print routine.

Printer Setup - Displays the Print Setup screen. The screen's contents will vary depending on the selected print device and the operating system.

Notes & Considerations

● The *Printer Dimensional Units* for the margin values are set on the *Other* data tab on the Report Options screen.

● Report contents may be printed to a "PDF" format if a third-party PDF printer is installed. To print to a PDF file, select the PDF printer name from the *Printer* list, then click the *Print* command button.

Print Settings - Short Form

GASWorkS uses two Print Settings screens - a "long" form and a "short" form. The "Short Form" is displayed when either the *Print* icon or the *Print* command button is clicked from one of the various data screens or a non-grid style report screen. The Print Settings screen allows various printer settings to be set by the User. The features associated with the screen are described as follows.

Displayed Items

Name - Displays the name of the font to be used when printing. Double-click in the data field to open the Font settings screen. Select the desired font, then click the *OK* command button.

Size - Displays the size of the font to be used when printing. Double-click in the data field to open the Font settings screen. Select the desired size, then click the *OK* command button. The size is expressed in printer points. One printer point is approximately equal to 1/72 of an inch.

Data Items

Bottom Margin - Specifies the distance from the bottom of the paper page to the bottom of the printed page.

Collate Pages - When selected, the pages are collated when multiple copies are printed.

Landscape Orientation - When selected, landscape orientation will be used when printing. When unselected, the pages will be printed using portrait orientation.

Left Margin - Specifies the distance from the left side of the paper page to the left side of the printed page.

Number Of Copies - Specifies the number of copies to be printed. Enter a value greater than zero (0).

Printer - Specifies the printer to use.

Right Margin - Specifies the distance from the right side of the paper page to the right side of the printed page.

Top Margin - Specifies the distance from the top of the paper page to the top of the printed page.

Command Buttons

Cancel - Closes the screen without proceeding with the Print routine.

Print - Proceeds with the Print routine.

Printer Setup - Displays the Print Setup screen. The screen's contents will vary depending on the selected printer and the configuration of the computer system.

Notes & Considerations

● The *Printer Dimensional Units* for the margin values are set on the *Other* data tab on the Report Options screen.

● The contents may be printed in a "PDF" format if a third-party PDF printer is installed. To print to a PDF file, select the PDF printer name from the *Printer* list, then click the *Print* command button.

Project Specifications

The Project Specifications screen allows predefined settings to be used when creating a new model, avoiding the need to set them individually. The Project Specifications screen is accessed by selecting the *New Project* item from the *File* menu. The features associated with the screen are described as follows.

Displayed Items

Background Image Files - Displays a list of the attached background image files to attach to the new model.

GASWorkS™ 10.1 — *Data Screens*

Data Items

Attach A Linked Customer Attribute File - When selected, a linked customer attribute file will be attached to the new model. The adjacent drop-down list which contains the available Attribute Database Definitions, will be enabled.

Attach A Linked Node Attribute File - When selected, a linked node attribute file will be attached to the new model. The adjacent drop-down list which contains the available Attribute Database Definitions, will be enabled.

Attach A Linked Pipe Attribute File - When selected, a linked pipe attribute file will be attached to the new model. The adjacent drop-down list which contains the available Attribute Database Definitions, will be enabled.

Default Settings - Specifies the default settings file to be associated with the new model. Either select a previously saved file or select "Default" to use the settings associated with the most previously open model.

Facility Settings - Specifies the facility settings file to be associated with the new model. Either select a previously saved file or select "Default" to use the currently saved "default" file.

Model Name - Specifies or displays the name of the new model. Enter a model name or click the *Select From Existing Models* command button to select an existing model Filename.

Pipe/Main Sizing Table - Specifies the Pipe Property Table used when calculating pipe sizes during the Solution routine.

Preference Settings - Specifies the preference settings file associated with the new model. Either select a previously saved file or select "Default" to use the currently saved "default" file.

Template Model - Specifies the name of an existing model file to use as a "template" for creating the new model. The dropdown list contains model files saved in the current Model Files path and the Settings subdirectory (subfolder) of the Support directory (folder). Either select a name from the list or select "Default" to use the currently saved "default" file.

Command Buttons

Cancel - Closes the screen without creating a new model or saving changes.

Continue - Closes the screen and creates a new model based on the specified settings.

Open - Opens a previously saved Project Specifications file (.gpj). When clicked, the File Selection screen is displayed.

Save - Saves the contents of the currently displayed specifications. When clicked, the File Selection screen is displayed. Once saved, the file may be opened and used with other models.

Select From Existing DXF Files - Displays the File Selection screen to allow the selection of an existing DXF image file to be used as a background image.

Select From Existing Models - Displays the Model Selection screen to allow the selection of an existing Model Name.

Notes & Considerations

● To attach a background image file, click the *Select From Existing DXF Files* command button.

● Only DXF-style background image files may be attached using the Project Specifications screen. To attach other file types of background images, use the Background Image Settings screen after the model has been created and opened.

● When one of the *Attach A Linked Attribute File* items is selected, an appropriately linked attribute file (database) file will need to be attached to the model. See the *Attribute Data* section in this manual for more information on working with linked attribute files.

● A Default Settings file may be created using the Default Data Values screen. See the *Default Data Values* topic in the *Data Screens* section for more information on creating and saving a file. If the "Default" item is selected for the *Default Settings* item, the values associated with the most recently open model will be used.

● A Facility Settings file may be created using the Facility Settings screen. See the *Facility Settings* topic in the *Data Screens* section for more information on creating and saving a file. If the "Default" item is selected for the *Facility Settings* item, the values associated with the most recently open model will be used.

● A Preference Settings file may be created using the *Save Current Settings* item from the *Preferences* submenu of the *File* menu. If the "Default" item is selected for the *Preference Settings* item, the values associated with the most recently open model will be used. The items associated with the Preference Settings include the Path Settings, Property Table assignments, and preference options.

Property Table Selection

The various Property Tables GASWorkS uses may be set using the Property Table Selection screen. The Property Table Selection screen is accessed by selecting the *Property Tables* item from the *Preferences* submenu of the *File* menu. The features associated with the screen are described as follows.

```
Property Table Selection...
┌─ Current Property Table Path... ──────────────────┐
│   C:\Program Files (x86)\GASWorkS 10\Properties\   │
│   ┌───────────────────────────────────────────┐   │
│   │              Change Path                  │   │
│   └───────────────────────────────────────────┘   │
└───────────────────────────────────────────────────┘
┌─ Select Desired Property Tables... ───────────────┐
│   Pipe:            [pipe              ▼]           │
│   Regulator:       [regulator         ▼]           │
│   Compressor:      [compressor        ▼]           │
│   Valve:           [valve             ▼]           │
│   Well:            [well              ▼]           │
│   Fittings:        [fitting           ▼]           │
│   ─────────────────────────────────────           │
│   Pipe Sizing Group:  [pipe_size_group ▼]          │
└───────────────────────────────────────────────────┘
┌──────────────────┐  ┌──────────────────┐
│      Cancel      │  │       Close       │
└──────────────────┘  └──────────────────┘
```

Displayed Items

Current Property Table Path - Displays the current location (path) of the Property Table files. The path may be changed by clicking the *Change Path Setting* command button or double-clicking on the current path label.

Data Items

Compressor - Specifies the current Compressor Property Table for the compressor values.

Fittings - Specifies the current Fittings Property Table for the fitting values.

Pipe - Specifies the current Pipe Property Table for the pipe values.

Pipe Sizing Group - Specifies the Pipe Sizing Group definitions in the Pipe Property Table.

Regulator - Specifies the current Regulator Property Table for the regulator values.

Valve - Specifies the current Valve Property Table for the valve values.

Well - Specifies the current Well Property Table for the well values.

Command Buttons

Cancel - Closes the screen without saving changes.

Change Path - Displays the Path Settings screen to allow the path (location) to be selected/set for each Property Table.

Close - Closes the screen and saves changes.

Notes & Considerations

● Only Property Tables in the current Property Table path (location) are displayed in the associated drop-down lists.

● When moving a model from one machine to another, the Property Tables may not be valid on the new machine. They may not contain the same information as the previous Property Tables. When moving a model to another machine, it is recommended that the model is first bundled using the *Bundle A Model* item from the *Bundle* submenu of the *File* menu, selecting the *Include Assigned Property Tables* option. Move the bundled file and unbundle it using the *Unbundle A Model* item from the *Bundle* submenu of the *File* menu - this will allow both the model and the Property Tables to be moved together.

Query Specifications

The Make Selection Set routine found in the various Data Reports, the Color-By-Query routine, and the various Flag Item routines allow the selection of data features based on User-specified selection criteria. These selection sets are created using the Query Specifications screen. The features associated with the screen are described as follows.

```
Query Specifications...

┌─ Find Records Where... ──────────────────────────────────┐
│                                                          │
│  Where...              Is...        To...                │
│  ┌──────────────────┐ ┌─────────┐ ┌──────────────────┐   │
│  │ Pipe Color     ▼│ │ =     ▼│ │ Default        ▼│   │
│  └──────────────────┘ └─────────┘ └──────────────────┘   │
│                                                          │
└──────────────────────────────────────────────────────────┘

┌─ Saved Specifications... ────────────────────────────────┐
│  ┌────────────────────────────────────────────────┐ ▼   │
│  └────────────────────────────────────────────────┘     │
│  ┌────────────────────────┐  ┌────────────────────────┐  │
│  │ Save Current Specification │ │ Delete Displayed Specification │ │
│  └────────────────────────┘  └────────────────────────┘  │
└──────────────────────────────────────────────────────────┘

┌────────────────────────┐  ┌────────────────────────┐
│   Perform Query        │  │       Close            │
└────────────────────────┘  └────────────────────────┘
```

Data Items

Is - Specifies the search operator to be used during the Query Selection. The search operator specifies the relation between the *Where* and *To* values.

Saved Specifications - Specifies a previously saved Query Specification to use for the current specification.

To - Specifies the value used as the check value during the Query Selection. The characteristics of this item will change depending on the selected *Where* item.

Where - Specifies which data item will be used as the search item during the Query Selection.

Command Buttons

Close - Closes the screen.

Delete Displayed Specification - Deletes the currently displayed specification in the *Saved Specifications* list.

Perform Query - Performs the query selection.

Save Current Specification - Saves the currently displayed specification in the *Find Records Where* data fields.

Notes & Considerations

● When creating a selection set in a report, the current selection set may be cleared by clicking the *Reset Selection Set* icon.

● See the *Query Features* topic in the *Miscellaneous* section for the meanings of the search operators.

Quick Pipe Sizing Specification

Sizing "tasks" may be combined into a sequence of tasks using the Quick Pipe Sizing Specification. The specification is manipulated using the Quick Pipe Sizing Specification screen. The screen is accessed by selecting the *Quick Size Specification* item from the *Edit* menu. The features of the screen are described as follows.

Displayed Items

Sequence - Displays the order in which the Facility Types will be processed (sized). For example, if the value of this item is "3," the pipes associated with the associated Facility Type will be sized third in the Pipe Sizing routine. Use the *Move Sequence* icon to change the order.

Data Items

Facility Type - Specifies the Facility Type for the associated sequence. The list's contents represent the Facility Types associated with the current model.

Minimum Pressure - Specifies the minimum pressure value for the associated sequence.

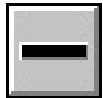**Pressure Units -** Specifies the dimensional units for the minimum pressure value.

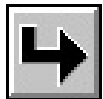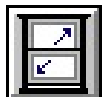**Sizing Method -** Specifies the sizing/processing method for the associated sequence.

Icons
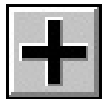
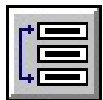**Add Sequence -** Adds a new sequence to the end of the specification.

Delete Sequence - Deletes the selected (highlighted) sequence.

Insert Sequence - Inserts a new sequence just before the selected (highlighted) sequence.

Maximize Screen Size - Increases the Quick Pipe Sizing Specification screen size to fill the GASWorkS display.

Move Sequence - Moves the selected (highlighted) sequence to a new location in the specification. When clicked, a message will be displayed to select the new location. Click the *OK* command button to clear the message, then select the desired location.
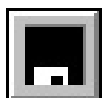
Open - Opens a previously saved Quick Pipe Sizing Specification file (.qwk). When clicked, the File Selection screen is displayed.

Print - Prints the contents of the specification. When clicked, the Print Settings screen is displayed.
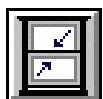
Reset Screen Size - Sets the Quick Pipe Sizing Specification screen's size to its "normal" dimensions and position.

Save - Saves the contents of the currently displayed specification. When clicked, the File Selection screen is displayed. Once saved, the file may be opened and used with other models.

Command Buttons

Cancel - Closes the screen without saving changes.

Close - Closes the screen and saves changes.

Run All Steps - Executes all of the sizing tasks listed in the specification in the order of their sequence number.

Run Individual Steps - Executes only the selected (highlighted) sizing task.

Notes & Considerations

● The Quick Pipe Sizing Specification may pre-define sizing specifications based on Facility Type. If a particular set of sizing options, values, and settings are commonly used to size pipes associated with a specific Facility Type, those values may be entered into the specification, saved, and used with the *Quick Size* command to quickly solve and size the matching pipes in the current model. Multiple facilities may be included in the specification. However, the model does not need to contain all the included Facility Types. Different specifications may be created for different model/project types. The specification may be particularly useful when sizing multiple pressure tier systems. For example, the first task/sequence might be to size the low-pressure portion of the system, then the medium-pressure portion, and then the high-pressure portion.

Report Options

The number of decimal places displayed by the various reports, data screens, and GDI Window, along with other settings, may be set using the Report Options screen. The screen is accessed by selecting the *Report Options* item from the *Report* menu. The features associated with the screen are described as follows.

Decimals Tab

Data Items

Number Of Decimals To Display - Specifies the number of decimal places to display for the associated data *Item* (parameter) type.

Other Tab

Data Items

Indicate Which Options To Include In Exceptions Report - When selected, the associated option will be included in the Exceptions Report.

> **Compressors With High Inlet Pressure -** Compressors with inlet pressures greater than the set pressure are reported.

> **Large Pressure Nodes -** Nodes with unusually high-pressure values are reported.

> **Low Flow Pipes -** Pipes with flows less than the Convergence Tolerance are reported.

> **Negative Pressure Nodes -** Nodes with pressure values less than zero (0) are reported.

> **Node Errors Greater Than Convergence Tolerance -** Nodes with a flow error greater than the Convergence Tolerance are reported.

Regulators With Back Flow - Regulators with flow towards the outlet node (away from the inlet node) are reported.

Regulators With Low Inlet Pressure - Regulators with inlet pressure values below their set pressure are reported.

Regulators With Low Outlet Pressure - Regulators with outlet pressure values below their set pressure are reported.

Zero Flow Pipes - Pipes with zero (no) flow are reported.

Zero Pressure Nodes - Nodes with zero (0) pressure values are reported.

Pressure Drop Per - Specifies the unit value to calculate the *Pressure Drop Per Unit Length* values.

Printer Dimensional Units - Specifies the dimensional units to apply to the print settings (used for the margins).

Sort All Reports When Opened - Indicates whether the Customer Data, Node Data, or Pipe Data Report are sorted when they are opened and how.

Command Buttons

Cancel - Closes the screen without saving changes.

Close - Closes the screen and saves changes.

Save As Default - Saves the current report settings as the default report options for creating a new model.
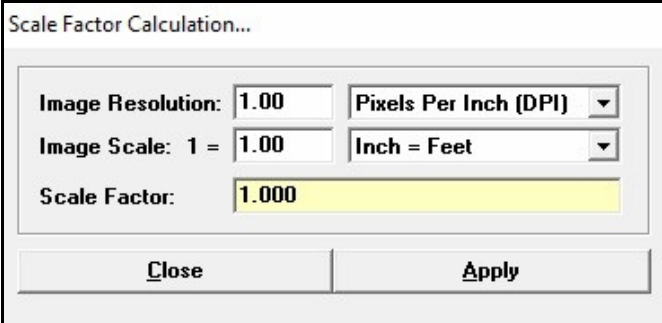
Notes & Considerations

● The value of the *Number Of Decimals To Display* indicates the number of digits displayed to the right of the decimal identifier (point) when displaying the associated data value.

● The *Pressure Drop Per Unit Length* value is calculated by dividing the product of the linear pressure drop and the length of the pipe segment by the *Pressure Drop Per* (Length) value.

● The Exception Report Options only apply when the *Include Exceptions In Solution Log* option is selected in the Solution Data options. The Exceptions Report may be used to troubleshoot a model that is not converging or as a quick double-check for model validity.

GASWorkS™ 10.1   *Data Screens*

Scale Factor Calculation

When a "raster" format file type is used as a background image file, and the *Calculate Scale Factor* command button on the Background Image Settings screen is clicked, the Scale Factor Calculation screen is displayed. The features associated with the screen are described as follows.

```
Scale Factor Calculation...

    Image Resolution:  1.00      Pixels Per Inch (DPI)  ▼

    Image Scale:  1 =  1.00      Inch = Feet            ▼

    Scale Factor:      1.000


        Close                         Apply
```

Data Items

Image Resolution - Specifies the resolution of the image contained in the selected background image file. Generally, the resolution is expressed in the number of pixels (dots) per inch.

Image Scale - Specifies the scale of the image contained in the selected background image file.

Calculated Items

Scale Factor - Displays the calculated Scale Factor value based on the image resolution and scale values.

Command Buttons

Apply - Closes the screen and applies the calculated *Scale Factor* value to the background image file on the Background Image Settings screen.

Close - Closes the screen without applying the calculated *Scale Factor* value.

Notes & Considerations

● For bitmap or raster-style images, the Scale Factor (Coordinate-To-Pixel Ratio) establishes the "scale" for the image. Bitmap and raster-style images comprise small "dots" called pixels. The scale tells the GDI how many *Coordinates* units are represented by one pixel in the file - the pixel's width in the *Coordinates* units. For example, the number of Feet-Per-Pixel. To determine this value, divide the original drawing or map scale by the image resolution in dots per inch (dots per centimetre). For example, the Coordinate-To-Pixel Ratio for a 1 inch = 2000 feet map scanned at 400 dots-per-inch would be: $2000 \div 400 = 5$ Feet-per-pixel.

● Bitmap and raster-style images include files ending with the following extensions:
- .bmp
- .jpg
- .pdf - PDF format files are not raster-type files; however, they are handled as raster-type files in GASWorkS.
- .tif

Solution Data

GASWorkS provides access to the various solution options and data values through the Solution Data screen. To access the screen, click the *Solve* icon from the lower-right corner of the GDI Window, select the *Solve Single Model* item from the *Analysis* menu, or use the *Solve Model* command. The features of the screen are described as follows.

Solution Data Tab

Data Items

Atmospheric Pressure - Specifies the atmospheric pressure method used to calculate the atmospheric pressure value for the associated node elevation. The atmospheric pressure is used to convert gauge pressure values to absolute pressure values as needed in the calculations. A description of the available methods may be found in the *Atmospheric Pressure Methods* topic in the *Appendix* section.

Compressibility - Specifies the compressibility method to be used in the solution. The Compressibility Factor compensates for the deviation of gas behavior from the ideal gas law. Generally, this deviation may be ignored for systems operating at pressures less than 200 psig. A description and discussion of the available methods may be found in the *Compressibility Methods* topic in the *Appendix* section.

Convergence Tolerance - Specifies the maximum acceptable node error for the solution. See the *Notes & Considerations* section for suggestions on an appropriate Convergence Tolerance.

Design Factor - Specifies a multiplication factor to be applied to the node loads during the solution. The value is expressed as a decimal. See the *Notes & Considerations* section for additional details.

Divergence Limit - Specifies the value of the maximum number of consecutive diverging or same value iterations allowed during the solution. If the number is exceeded, the solution will be terminated. See the *Notes & Considerations* section for additional details.

Diversity - Specifies the diversity method to calculate the IGE Diversity calculations during the solution. When a method is selected, this will cause the load associated with "non-diversified" customers to be applied to the downstream node of the supplying main, regardless of the node application setting. More information may be found in the *Diversified Customer Loads* topic in the *Appendix* section.

Maximum Iterations - Specifies the maximum permissible number of iterations the solution can make before ending. See the *Notes & Considerations* section for additional details.

Solution Options -

> **Adjust Compressor Control Pressures -** When selected, the compressor control pressures will be adjusted after/between solution attempts.

> **Adjust Regulator Outlet Pressures -** When selected, the regulator outlet pressures will be adjusted after/between solution attempts.

> **Calculate Gas Mixing -** When selected, the gas property values will be calculated based on the mixed flow rates.

> **Calculate Temperature Values -** When selected, the average temperature will be calculated for each node where the *Temperature Known* item is "No."

> **Display Favorite Results List Upon Completion -** When selected, the Favorite Results List in the Data Panel will be displayed upon completion of the Solution routine.

> **Flag Large Error Node Upon Completion -** When selected, a flag symbol will be placed at the node's location with the largest error. The flag will be placed after the solution routine is finished.

> **Ignore Elevation Differences -** When selected, the node elevation differences will be ignored when calculating the node "hydrostatic" pressure values. However, the correct atmospheric pressure values will be used in the Absolute Pressure calculation.

Include Exceptions In Solution Log- When selected, a check for unusual conditions will be performed when the data is saved after the solution. Any issues that are found are referred to as "exceptions" and are included in the Solution Log. The exceptions included in the check are controlled by the Report Options screen.

Reset Unknown Node Pressures To Zero - When selected, the "unknown" pressure values will be set to zero (0) before solving the model. This option is advised when using the *Calculate Pipe Sizes (Optimize)* option or diversity options.

Update Color Coding - When selected, the color-by-range color coding will be updated upon completion of the solution. The current range and color criteria are used to update the GDI Image.

Pipe Sizing Control Tab

See the *Notes & Considerations* section or the *Pipe Sizing (Optimization) Features* topic in the *Miscellaneous* section for additional information on the Pipe Sizing routine and the effect of the various data items and options on the sizing process.

```
Solution Data...
 ┌─────────────────────────────────────────────────────────────────────────────┐
 │  Solution Data    │  Pipe Sizing Control  │  Other Settings  │  Model Notes  │
 │ ┌─ Limit/Check Values... ──────────────────────────────────────────────────┐ │
 │ │  Condition Nodes:    [              ▼]  Condition Pressure: [      ]  Psi │ │
 │ │  System Pressures:   [10    ] [Psi      ▼]                                │ │
 │ │  Pressure Values Are:  (•) Minimums    ( ) Maximums                       │ │
 │ │  Maximum Velocity:   [40    ]      Feet/sec                               │ │
 │ └──────────────────────────────────────────────────────────────────────────┘ │
 │ ┌─ Other Settings... ──────────────────────────────────────────────────────┐ │
 │ │  Pass Limit:         [50    ]      Number Of Passes                       │ │
 │ │  Path Processing:    [Flow-Pressure Drop Processing ▼]  Optimize By:  (•) Size  ( ) Cost │ │
 │ │  Facility Type:      [All Facility Types            ▼]  Reset Diameters To Minimum Size ☑ │ │
 │ │  Pipe Sizing Table:  [pipe                          ▼]                    │ │
 │ │              [        Solve & Calculate Pipe Sizes        ]               │ │
 │ └──────────────────────────────────────────────────────────────────────────┘ │
 │ ┌──────────────┐   ┌──────────────┐        ┌──────────────┐                  │
 │ │   Cancel     │   │    Close     │        │    Solve     │                  │
 │ └──────────────┘   └──────────────┘        └──────────────┘                  │
 └─────────────────────────────────────────────────────────────────────────────┘
```

Data Items

Condition Nodes - Specifies up to ten "condition" node names. If no condition nodes are identified, only the overall system pressure will be used.

Condition Pressure - Specifies the required pressure value for the associated condition node location.

Facility Type - Specifies the Facility Type to include in the Pipe Sizing routine.

Maximum Velocity - Specifies the maximum allowable velocity value.

Optimize By - Indicates whether the pipe size or pipe cost will be used as the selection basis during the Pipe Sizing routine. When the *Size* option is selected, smaller pipe sizes will be used before larger pipe sizes. When the *Cost* option is selected, lower-cost pipes will be used before higher-cost pipes.

Pass Limit - Specifies the number of "passes" or attempts that the Pipe Sizing routine is allowed to execute. If the limit is reached before the routine is complete, the routine will be stopped, and the current pipe sizes will be saved. To restart the Pipe Sizing routine after the Pass Limit has been reached, re-solve the model, remembering to unselect the *Reset Diameters To Minimum Size* option.

Path Processing - Specifies the type of path processing method that will be used during the Pipe Sizing routine. During the Pipe Sizing routine, the lowest pressure node is found (when *Pressure Values Are* "Minimums" is selected). Then, a path is determined along the pipes feeding into the low-pressure node, back to the supply source. The size of the pipes along this path is increased. The *Path Processing* item specifies how the path is created and how the pipes are selected for a size increase.

Pipe Sizing Table - Specifies the Pipe Property Table to be used when calculating pipe sizes by the Pipe Sizing routine.

Pressure Values Are - Indicates whether the check (Condition and System) pressure values represent maximum or minimum values.

Reset Diameters To Minimum Size - When selected, the size for all pipes where the *Allow Sizing* option is "Yes" will be set to the minimum value before starting the Pipe Sizing routine.

System Pressures - Specifies the value of the overall system pressure requirement.

Other Settings Tab

Data Items

Ambient Temperature - Specifies the temperature outside the "influence zone." This item is only displayed when the *Convection & Conduction* or *Overall Transfer Coefficient* Calculation Methods are selected.

Base Pressure - Specifies the pressure basis for the volume measurement.

Base Temperature - Specifies the temperature basis for the volume measurement.

Calculation Method - Specifies the method to calculate the heat exchange between the flowing gas and the surrounding environment. The available methods are described in the following *Notes & Considerations* section.

Diameter Path Factor - Specifies the diameter factor (exponent) to use when an applicable *Path Processing* method is used during the Pipe Sizing routine. This item will only be enabled when the *Flow Processing Only* Path Processing method is selected.

Environment Conductivity - Specifies the thermal conductivity of the environment within the influence zone. This item is only displayed when the *Convection & Conduction* Calculation Method is selected.

Include Joule-Thomson Effect For Pipes - When selected, the Joule-Thomson effect (cooling due to pressure drop) will be estimated for each pipe segment.

Influence Factor - Specifies the influence zone size of the environment. The pipe outside diameter is multiplied by the specified factor to determine the thickness of the influence zone thickness of the environment surrounding the pipe. This item is only displayed when the *Convection & Conduction* Calculation Method is selected.

Insulation Conductivity - Specifies the thermal conductivity of the insulating material. If no insulation is present, enter a value of zero (0). This item is only displayed when the *Convection & Conduction* Calculation Method is selected.

Insulation Thickness - Specifies the thickness of any insulation applied to the pipe's exterior. If no insulation is present, enter a value of zero (0). This item is only displayed when the *Convection & Conduction* Calculation Method is selected.

Length Path Factor - Specifies the length factor (exponent) to use when an applicable *Path Processing* method is used during the Pipe Sizing routine. This item will only be enabled when the *Flow-Pressure Drop Processing, Flow Processing Only,* or the *Sum Of Pressure Loss* Path Processing methods are selected.

Lower Dampening Factor - Specifies the lower dampening factor for the solution. The influence of the lower dampening factor is described in the *Notes & Considerations* section.

Pipe Conductivity - Specifies the thermal conductivity of the pipe material. This item is only displayed when the *Convection & Conduction* Calculation Method is selected.

Pressure Drop Path Factor - Specifies the pressure drop factor (exponent) when an applicable *Path Processing* method is used during the Pipe Sizing routine. This item will only be enabled when the *Flow-Pressure Drop Processing* or the *Sum Of Pressure Loss* Path Processing methods are selected.

Row Reduction Limit - Specifies the row reduction limit for the solution. The influence of the row reduction limit is described in the *Notes & Considerations* section.

Transfer Coefficient - Specifies the value of the overall heat transfer coefficient. This item is only displayed when the *Overall Transfer Coefficient* Calculation Method is selected.

Wall Temperature - Specifies the temperature of the pipe wall. This item is only displayed when the *Convection Only* Calculation Method is selected.

Model Notes Tab
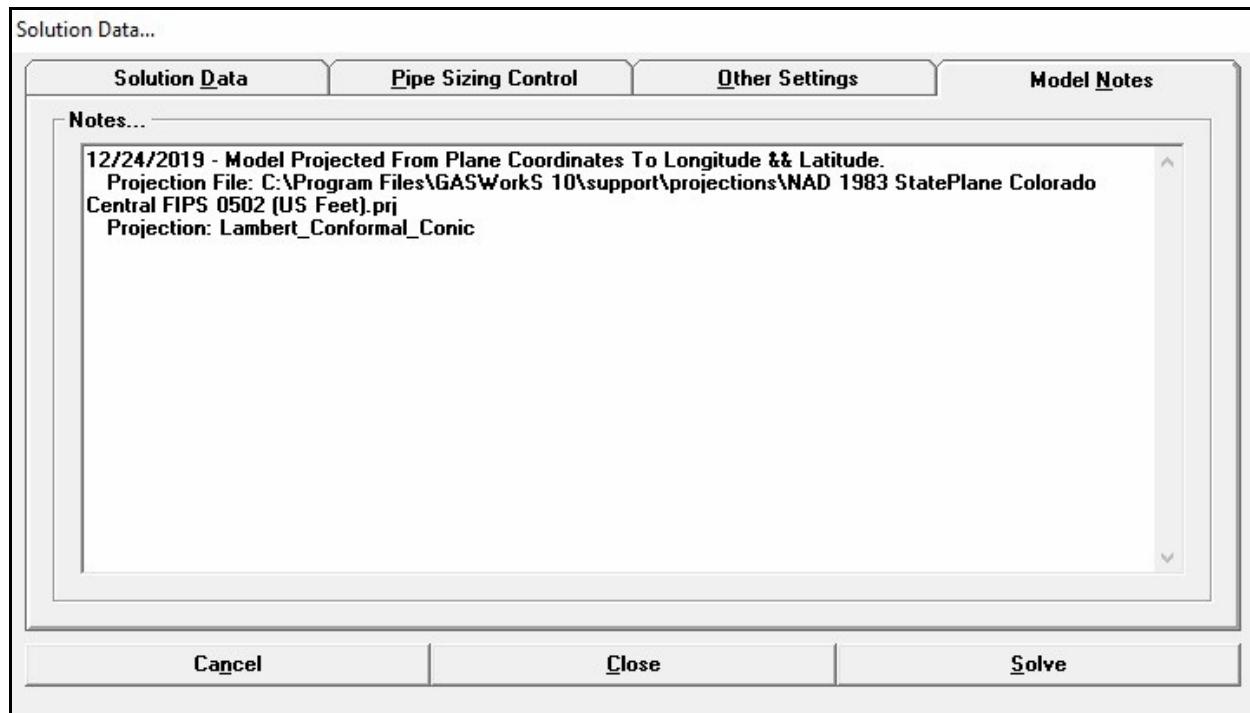
Data Items

Notes - Displays the Model Notes.

Command Buttons

Calculate - Allows calculation of the Design Factor. When selected, the Design Factor Calculation screen is displayed. Enter the desired values, click the *Calculate* command button, then click the *Apply* command button to apply the *Calculated Design Factor*. This item is displayed on the *Solution Data* tab.

Cancel - Closes the screen without proceeding with the Solution routine or saving changes.

Close - Closes the screen and saves changes without executing the Solution routine.

Reset - Resets the default *Advanced Solution Parameter* values. This command button is displayed on the *Reset* data tab.

Solve - Saves the settings and executes the Solution routine.

Solve & Calculate Pipe Sizes - Executes the Solution routine and calculates the pipe sizes. The command button serves the same function as selecting the *Calculate Pipe Sizes (Optimize)* solution option and then clicking the *Solve* command button. This command button is displayed on the *Pipe Sizing Control* data tab.

Notes & Considerations

● The Design Factor is used to adjust the node loads in the model. Depending on the individual node settings, the "known" node loads are multiplied by the Design Factor to calculate the load values used in the analysis. The Design Factor is expressed as a decimal. For example, to enter a Design Factor representing fifty percent, type 0.5. The Design Factor is only applied to nodes where the *Adjust Base Load* node option is "Yes." The Design Factor is not applied to the *External Load* values unless the *Adjust External Load* node option and the *Adjust Load* customer option are set to "Yes."

● The Maximum Iteration value limits the number of times that the Solution routine will attempt to reach a solution within the specified Convergence Tolerance. The number of iterations required to reach a solution varies with each model and with the value of the Convergence Tolerance. If a solution is not reached within the specified number of iterations, a message will be displayed stating that the solution has not been completed. The solution may be rerun, and the maximum number of iterations may be increased as required. However, generally, if a model cannot solve in less than thirty iterations, the model likely contains data errors that prevent the solution from being achieved. Run the Data Check routine and /or check the model configuration around the node or nodes with the highest flow error values.

● The Convergence Tolerance represents the maximum allowable imbalance of the solution results. If a solution is attained for a Convergence Tolerance of 0.01 mcfh, it may be said that the solution results are within +/- 0.01 mcfh of the exact solution for that particular model. The value of the Convergence Tolerance is generally determined based on the magnitude of the total expected system flows. The acceptable Convergence Tolerance for a particular model depends on the intended use of the results. For very "tight" results, use a Convergence Tolerance of about one-hundredth of one percent (one-ten-thousandth) of the total expected system flow. For example, if the total system flow is expected to be 100 mcfh, the Convergence Tolerance might be 0.01 mcfh. For less accurate results, use a Convergence Tolerance of about one percent of the total expected system flow. See the *Convergence Problems* topic in the *Miscellaneous* section for suggestions on handling various convergence problems and solution errors.

● The Divergence Limit identifies the number of consecutive diverging or equivalent error iterations allowed before the Solution routine is terminated. During the Solution routine, node errors normally decrease as the solution progresses. Sometimes, when solving a model, the node errors might increase for a few iterations and then begin to decrease again. However, when trying to solve a model containing a data or configuration error or when a model is "running out of gas," the node errors may continue to increase or stay the same, never allowing the solution to reach a state of balance. The Divergence Limit causes the Solution routine to be automatically terminated when this occurs.

● The base pressure and base temperature are used to establish the basis for a "standard" volume (for example, a standard cubic foot or cubic metre) of gas as used by the current model.

● The Row Reduction Limit is used to modify the calculated adjustment for large node errors. The modification will be applied to errors greater than the Row Reduction Limit multiplied by the Convergence Tolerance. The Row Reduction Limit generally does not influence the quality of the solution. However, it can prevent a solution from being reached if improperly chosen. Enter a large number (for example, 10000) to disable the Row Reduction Limit. Generally, the factor should be entered as zero (0).

● The Lower Dampening Factor is used to stop adjustment of certain node pressures. Often, several nodes in the model are balanced, while only a few nodes require further adjustment. The nodes where the pressure adjustment value is less than the current pressure value multiplied by the Lower Dampening Factor will not be adjusted any further. The Lower Dampening Factor can influence the quality of the solution. However, if the factor is properly chosen, it can decrease the number of iterations required to reach a solution, with only a negligible influence on the overall quality of the solution. Generally, the Lower Dampening Factor should be set to zero (0).

● The *Heat Exchange Parameters* Calculation Methods are described as follows:

> *Convection & Conduction* - This method is useful when considering a buried pipe that may or may not have insulating material applied to it. Convection between the gas stream and the pipe wall, conduction through the pipe wall, insulation, and soil (environment) are considered. An environmental influence zone may be specified using the Influence Factor. If no insulation is present, the insulation thickness may be set to zero (0).

> *Convection Only* - This method considers convection heat transfer between the gas stream and the pipe wall. This method would be useful when the pipe is exposed to the atmosphere, submerged in a liquid, or buried in soil with a high transfer rate - locations where the surrounding environment can act as a large heat sink and maintain a constant wall temperature on the pipe.

> *Overall Transfer Coefficient* - This method is useful when the value of the overall heat transfer coefficient is known or may be estimated. The heat transfer is based on the specified coefficient.

● During the Pipe Sizing routine, certain pressure requirements are checked to determine if the pipes are properly sized. These pressure requirements may be set in two manners. A pressure may be specified at a particular location in the system (a condition node), and an overall pressure may be set for the entire model. See the *Pipe Sizing (Optimization) Features* topic in the *Miscellaneous* section for additional information on using the Pipe Sizing routine and the effect of the various data items and options on the sizing process.

● To enter or revise the Model Notes, type them directly into the edit box. The Model Notes may consist of a description of the model and analysis. Special characters may be used. See the *Model Notes* topic in the *Data Screens* section for additional details.

● If the *Adjust Regulator Outlet Pressures* option is selected, the outlet pressure of any included regulator will be checked against the entered set pressure value, and the following actions will be taken.

• If the pressure is set as unknown for the outlet node, and the calculated pressure for the outlet node is higher than the set pressure, the regulator will be "turned off," and the solution routine will be repeated.

• If the pressure is set as unknown for the outlet node, and the calculated pressure for the inlet node is lower than the set pressure plus the associated minimum differential pressure, the outlet pressure is calculated based on the associated valve factor and minimum differential pressure values, and solution routine will be repeated.

• If the pressure is set as unknown for the outlet node, and the calculated pressure for the inlet node is greater than the set pressure plus the associated minimum differential, the outlet pressure of the regulator will be calculated based on the associated droop value, and the solution routine will be repeated.

• If the pressure is set as known for the outlet node, no action is taken.

• In the above actions, when the outlet pressure is calculated, the solution routine will be repeated until the change in the outlet pressure between subsequent solutions is less than a preset tolerance or the number of iterations exceeds the *Maximum Iteration* value.

● If the *Adjust Compressor Control Pressures* option is selected, the pressure of the "control" node of any included compressor will be checked against the entered set pressure value, and the following actions will be taken.
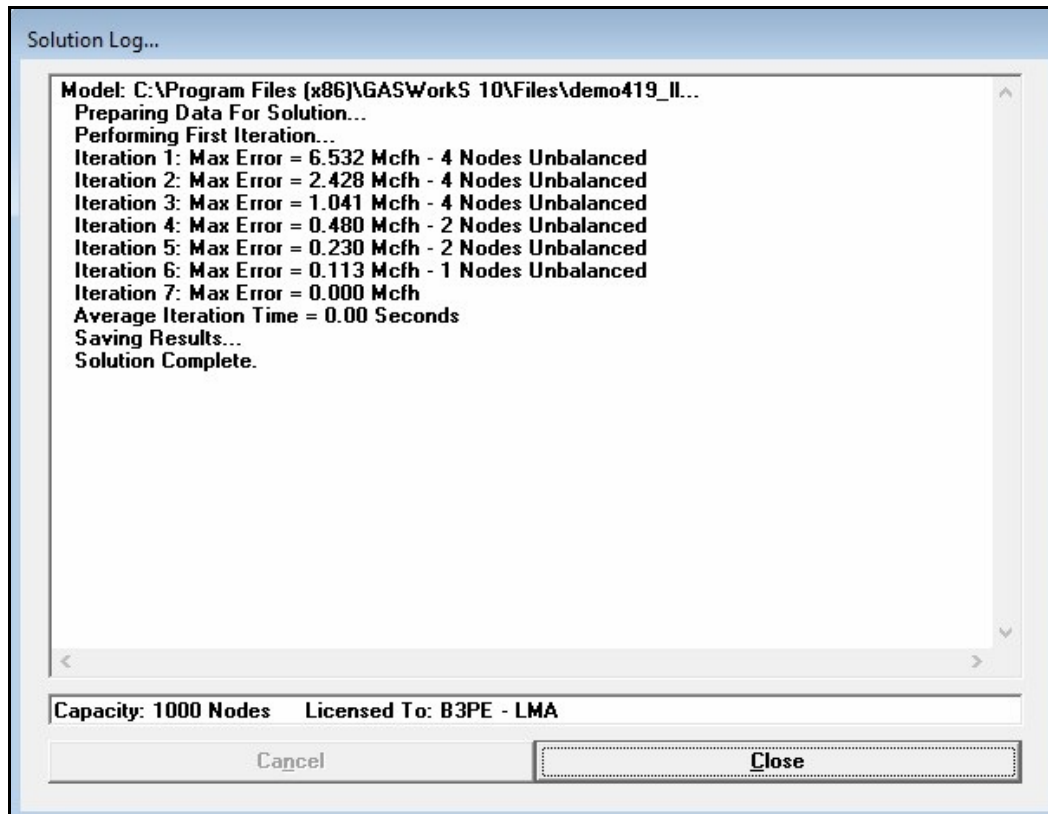
• If the control node is downstream (outlet), the pressure is set as unknown for the outlet node, and the calculated pressure for the outlet node is higher than the set pressure, the compressor will be "turned off," and the solution routine will be repeated.

• If the control node is downstream (outlet), the pressure is set as unknown for the outlet node, and the calculated pressure for the inlet node is higher than the set pressure, the compressor will be "bypassed," and the solution routine will be repeated.

• If the control node is upstream (inlet), the pressure is set as unknown for the inlet node, and the calculated pressure for the outlet node is lower than the set pressure, the compressor will be "turned off," and the solution routine will be repeated. However, an unsolvable condition may be created.

• If the control node is upstream (inlet), the pressure is set as unknown for the inlet node, and the calculated pressure for the inlet node is lower than the set pressure, the compressor will be "turned off," and the solution routine will be repeated. However, an unsolvable condition may be created.

• If the pressure is set as known for the control node, no action is taken.

• In the above actions, when the pressure is calculated, the solution routine will be repeated until the change in the calculated pressure between subsequent solutions is less than a preset tolerance or the number of iterations exceeds the *Maximum Iteration* value.

Solution Log

During the solution of a model, GASWorkS will display the Solution Log. The Solution Log allows control of the solution and viewing of the solution's progress. The Solution Log is displayed when a solution of the model is executed. The features associated with the screen are described as follows.

```
Solution Log...

Model: C:\Program Files (x86)\GASWorkS 10\Files\demo419_II...
   Preparing Data For Solution...
   Performing First Iteration...
   Iteration 1: Max Error = 6.532 Mcfh - 4 Nodes Unbalanced
   Iteration 2: Max Error = 2.428 Mcfh - 4 Nodes Unbalanced
   Iteration 3: Max Error = 1.041 Mcfh - 4 Nodes Unbalanced
   Iteration 4: Max Error = 0.480 Mcfh - 2 Nodes Unbalanced
   Iteration 5: Max Error = 0.230 Mcfh - 2 Nodes Unbalanced
   Iteration 6: Max Error = 0.113 Mcfh - 1 Nodes Unbalanced
   Iteration 7: Max Error = 0.000 Mcfh
   Average Iteration Time = 0.00 Seconds
   Saving Results...
   Solution Complete.


Capacity: 1000 Nodes     Licensed To: B3PE - LMA

        Cancel                          Close
```

Displayed Items

GASWorkS Information - Displays the node capacity and the name of the entity to which the GASWorkS software is licensed.

Solution Log - Displays the Solution Log. To browse the display, adjust the vertical scroll bar adjacent to the display area.

Command Buttons

Cancel - Terminates the Solution routine. However, depending on the current solution activities, the process may not terminate immediately. This button is disabled after the solution is complete.
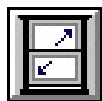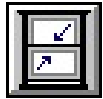
Close - Closes the Solution Log. This button is disabled during the Solution routine.

Notes & Considerations

● If the *Include Exceptions In Solution Log* solution option is selected before executing the Solution routine, any exceptions found while saving the solution results will be reported at the end of the Solution Log. If there are more exceptions than may be contained in the Solution Log, the reported exceptions will be truncated. An end-of-section message will be reported at the end of the exceptions section.

● The contents of the Solution Log may be printed by selecting the *Solution Log* item from the *Report* menu.

● If exceptions are listed, and the GDI is open, close the Solution Log, then reopen the screen using the *Solution Log* item from the *Report* menu. Then click on an exception item to flag and zoom to the location of the associated feature in the model.

GASWorkS™ 10.1   *Data Screens*

Solution Log Report

During the solution, a log of the solution activities will be created. The log is maintained in a report known as the Solution Log Report. The Solution Log Report may be viewed or printed by selecting the *Solution Log* item from the *Report* menu. The features associated with the report are described as follows.
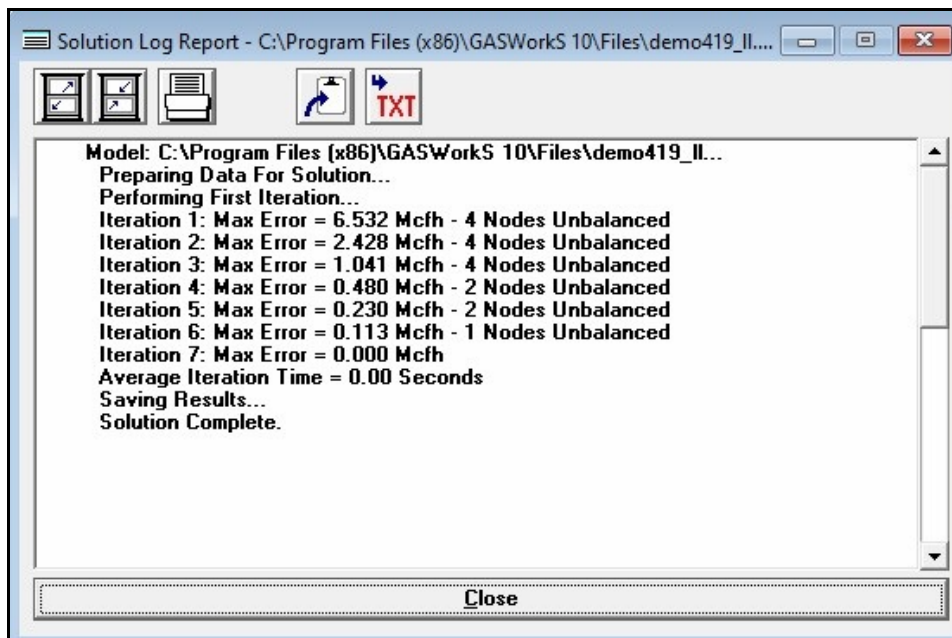
Displayed Items

Solution Log Report - Displays the Solution Log Report. To browse the display, adjust the vertical scroll bar adjacent to the display area.

Icons

Copy To Clipboard - Copies the contents of the Solution Log Report to the Windows Clipboard. Once copied, the contents may be pasted into another application.

Maximize Screen Size - Increases the Solution Log Report's size to fill the GASWorkS display.

Print - Prints the contents of the Solution Log Report. When clicked, the Print Settings screen is displayed.

Reset Screen Size - Sets the Solution Log Report's size to its "normal" dimensions and position.
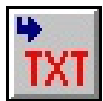
Save To Text File - Saves the data of the Solution Log Report contents to a text file (.txt). When clicked, the File Selection screen is displayed.
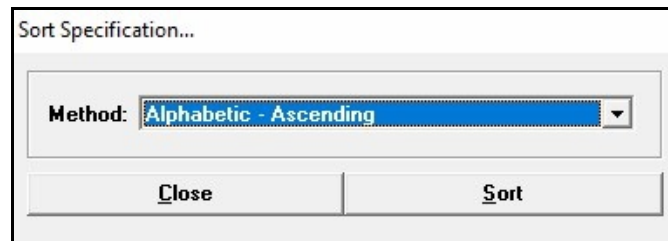
Command Buttons

Close - Closes the Solution Log Report.

Notes & Considerations

● If the *Include Exceptions In Solution Log* solution option is selected before the execution of the solution, any exceptions found while saving the solution results will be reported at the end of the Solution Log. If there are more exceptions than may be contained in the log, the reported exceptions will be truncated. An end-of-section message will be reported at the end of the exceptions section.

● If exceptions are reported, and the GDI is open, click on an item in the report to flag and zoom to the associated feature. When selected, the item will be displayed in "red" text until the report is closed.

Sort Specification

The Sort routine provides a variety of sorting options for reordering the pipes and nodes in the model data files. The sort features are controlled through the Sort Specification screen. The screen is accessed by selecting the *Sort Model* item from the *Miscellaneous* submenu of the *Utilities* menu. The features associated with the screen are described as follows.

Data Items

Method - Specifies how to sort the pipes and nodes in the model data files.

Command Buttons

Close - Closes the screen.

Sort - Sorts the node and pipes in the model data by the selected method.

Notes & Considerations

● To save the sorted model order, save the model.

System Summary Report

The System Summary Report includes several specialized report sections that may be selected by the User. These sections may include a summary of the regulator and compressor data, customer, node, and pipe data, supply nodes, system components, and other items. The User may indicate which sections to include in the report using the System Summary Options screen. The report is accessed by selecting the *System Summary* item from the *Report* menu. The features associated with the report are described as follows.
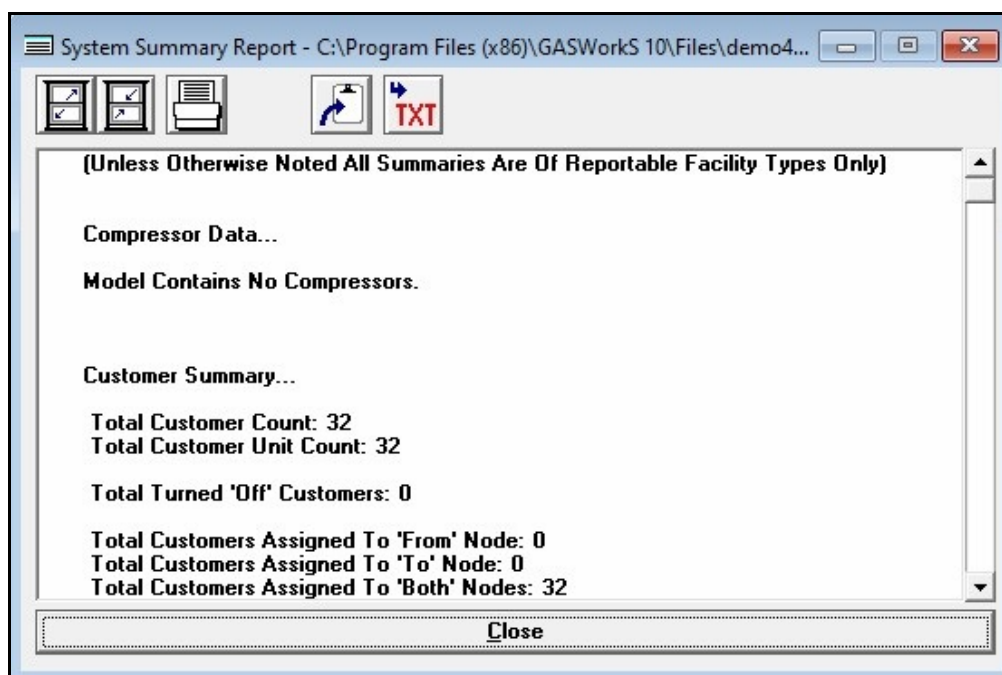
Displayed Items

System Summary Report - Displays the System Summary Report. To browse the display, adjust the vertical scroll bar adjacent to the display area.

Icons

 Copy To Clipboard - Copies the contents of the System Summary Report to the Windows Clipboard. Once copied, the contents may be pasted into another application.
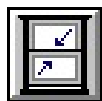
Maximize Screen Size - Increases the System Summary Report's size to fill the GASWorkS display.

Print - Prints the contents of the System Summary Report. When clicked, the Print Settings screen is displayed.
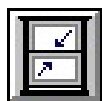
Reset Screen Size - Sets the System Summary Report's size to its "normal" dimensions and position.

Save To Text File - Saves the data of the System Summary Report contents to a text file (.txt). When clicked, the File Selection screen is displayed.

Command Buttons

Close - Closes the System Summary Report.

Notes & Considerations

● The *System Summary Options* topic in the *Data Screen* section provides additional information about the System Summary Report contents.

● The dimensional units of certain displayed items are automatically set. The units for these items are summarized in the following table:

| *Displayed Value*	*Section*	*Dimensional Units*
Cost	Pipe Summary	None
Differential Pressure	Regulator Data	Set to the *Diff P Units* of the associated record in the Regulator Property Table.
Fuel Usage Rate	Compressor Data	Set to the *Fuel Units* of the associated record in the Compressor Property Table.
Length	Pipe Summary	Set to the *Length Units* of the first reportable pipe in the model.
Minimum Differential Pressure	Regulator Data	Set to the *Diff P Units* of the associated record in the Regulator Property Table.

Displayed Value	*Section*	*Dimensional Units*
Power Requirement	Compressor Data	Set to the *Power Units* of the associated record in the Compressor Property Table.
Total Flow In-To The System	System Summary	Set to the *Load Units* of the first reportable node in the model.
Total Flow Out-Of The System	System Summary	Set to the *Load Units* of the first reportable node in the model.

● The Estimated Valve Coefficient calculated in the *Regulator Data* section represents the following parameter:

Equation	*Estimated Valve Coefficient*
American	C Factor - Regulator Constant
American AFV Grove 900 ISA-75.01 Mokveld	Cv - Valve Capacity Factor
Rockwell	Fv Factor - Regulator Constant
Fisher Mooney Pietro Fiorentini Universal	Cg (Regulating) - Gas Sizing Coefficient
Equimeter Itron	K Factor - Regulator Constant
Grove 80	Cp - Wide Open Valve Factor
Table	Not Calculated

System Summary Options

The System Summary Report includes several specialized report options. The User may indicate which sections to include in the System Summary Report. The System Summary Options screen is accessed by selecting the *System Summary* item from the *Report* menu. The features associated with the screen are described as follows.
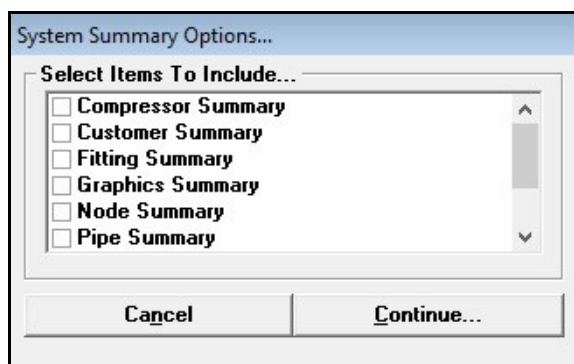
Data Items

Compressor Summary - When selected, the compressor data will be included in the System Summary Report. The compressor section reports various data and results about the compressors found in the system.

Customer Summary - When selected, certain customer data will be included in the System Summary Report. The customer section reports various data about the customer features found in the system.

Node Summary - When selected, certain node data will be included in the System Summary Report. The node section reports various data about the nodes found in the system.

Pipe Summary - When selected, certain pipe data will be included in the System Summary Report. The pipe section reports pipe footage, diameter values, and other data values grouped by diameter value.

Regulator Summary - When selected, the regulator data and results will be included in the System Summary Report. The regulator section reports various data about the regulators found in the system.

Supply Nodes Summary - When selected, a summary of the supply nodes will be included in the System Summary Report. The supply nodes section lists each node with a positive load value, along with its load and pressure values.

System Summary - When selected, a summary of certain system values will be included in the System Summary Report. The system summary section reports the number of nodes, pipes, known and unknown values, and total flow into and out of the system.

Command Buttons

Cancel - Closes the screen without creating the System Summary Report.

Continue - Displays the System Summary Report.

Notes & Considerations

● The System Summary Report provides a summary of the various data items contained in the model. The report can include various User-specified items. The available options are described in the following table:

Option	*Features*
Compressor Summary	• Calculates the compression ratio, the estimated outlet temperature, the estimated power requirement, the estimated fuel usage, the total power, and the total fuel. • Displays the From Node and To Node, the compressor type, the flow rate, the flow mode, the inlet temperature, the rated power, and the fuel usage rate.
Customer Summary	Displays the total customer count; the total customer unit count, the number of turned "off" customers, the "From Node," "To Node," and "Both Nodes" assignments, the total unadjusted load into and out of the system, the number of branch customers, header customers, and polyline customers, the number of customers with unassigned service line coordinates, and the number of deleted customers.
Node Summary	Displays the total node count, the connection node count, the control fitting node count, and the valve node count.
Pipe Summary	Displays the Size/Type Code, the inside diameter, the pipe count, the length, the cost, the total pipe count, the total length, and the total cost.
Regulator Summary	• Calculates the estimated valve coefficient, the differential pressure, the outlet temperature, and the coefficient ratio. • Displays the From Node and To Node, the regulator type, the flow rate, the flow mode, the inlet pressure, the outlet pressure, the differential pressure, the minimum differential pressure, the inlet temperature, the outlet temperature, the estimated valve coefficient, the published valve coefficient, and the coefficient ratio.

Option	*Features*
Supply Nodes Summary	• Lists all of the nodes with a positive load value. • Calculates the percentage of the total flow rate and the total of all flow rates. • Displays the node name, the flow rate, and the pressure.
System Summary	• Displays the total customer, node, and pipe count. • Displays the number and total of "known" and "unknown" loads, pressures, temperatures, and gas properties. • Displays the number of turned "off" nodes and the number of nodes connected only to turned "off" pipes (effectively rendering the nodes as turned "off"). • Calculates the base and external total flow (load) into and out of the system. • Displays the element summary and the facility summary. • Displays miscellaneous model and non-model data summary.

GRAPHIC DATA INTERFACE (GDI)

GASWorkS™ 10.1   *GDI*

This section provides instructions for using the GASWorkS Graphic Data Interface (GDI) window. A substantial number of the GASWorkS features and commands are accessed through the GDI Window. Each of these features, along with the associated data screens, is described in this section.

The GDI provides graphical access to the model data. The GDI Window may create a graphical representation of the model, which may be used to view, edit, and manipulate the GASWorkS model data. The GDI is not intended to be a "CAD" system; however, it does provide some CAD-type features. For more sophisticated graphic needs, we recommend using a true drafting/design, mapping, or GIS application. Data from these applications may then be shared with GASWorkS using one of the many Import routines.

The Help Guide provides examples and how-to tips for performing various tasks using the GDI. The remainder of the section describes the various features associated with the GDI.

GDI Window Description

The GDI features are accessed through the Graphic Data Interface (GDI) window. The features associated with the GDI Window and its layout are described in the following sections.
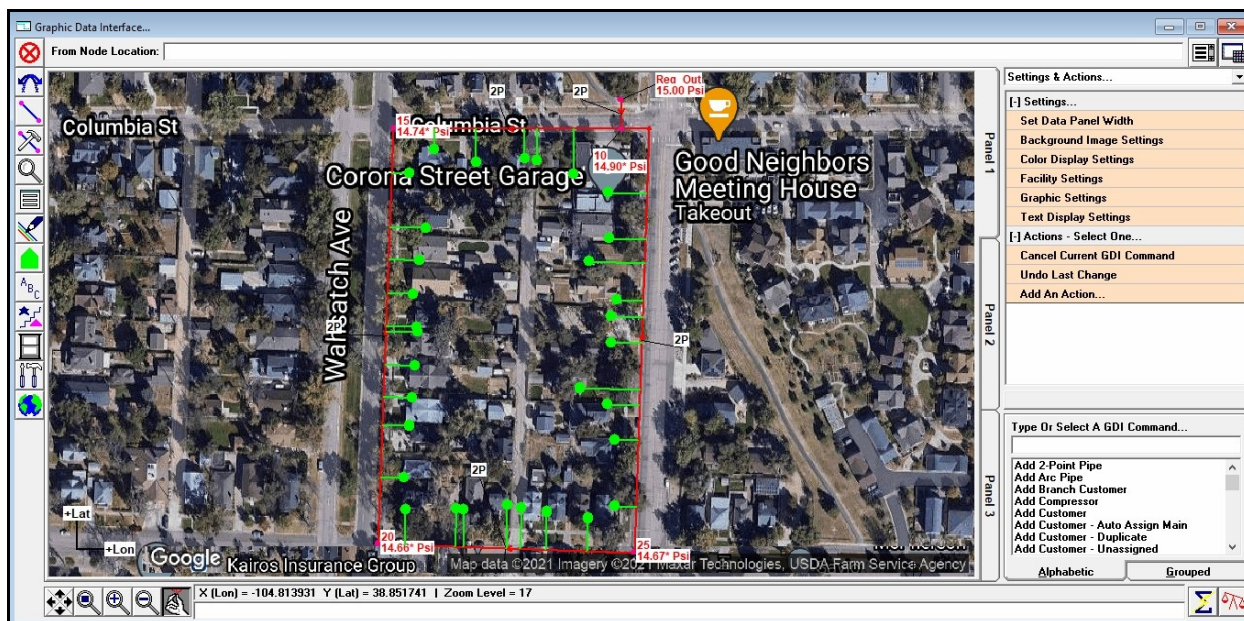
With the exception of the GDI Command Line, the features associated with the GDI may only be accessed using the mouse. If your computer system is not equipped with a mouse or compatible pointing device, you will not be able to use the GDI.

GASWorkS™ 10.1 *GDI*

The GDI contains several functional areas. The left side of the GDI contains a group of icons representing the GDI Toolbars *(Item A)*. A "fly-out" toolbar will be displayed when the mouse cursor is passed over each toolbar icon. The various GDI Window Icons are grouped by functionality on the specific GDI Toolbars. For example, icons for changing the scale and position of the displayed GDI Image are grouped on the *View Controls Toolbar*. Many GDI Commands are accessed through the GDI Icons displayed on the GDI Toolbars. At the same time, others can only be accessed through the GDI Command List or by typing an abbreviation on the GDI Command Line. The meaning and function of each of the GDI Commands and GDI Icons are described later in this section.

The central portion of the screen contains the GDI Display. The GDI Display is the screen portion where the model image is displayed and manipulated. The image's appearance in the display may be modified by panning (moving right or left or up or down) and zooming (enlarging or reducing) the image. Additionally, the color, size, and style of its features displayed in the GDI Display may be manipulated by the User.

Item A
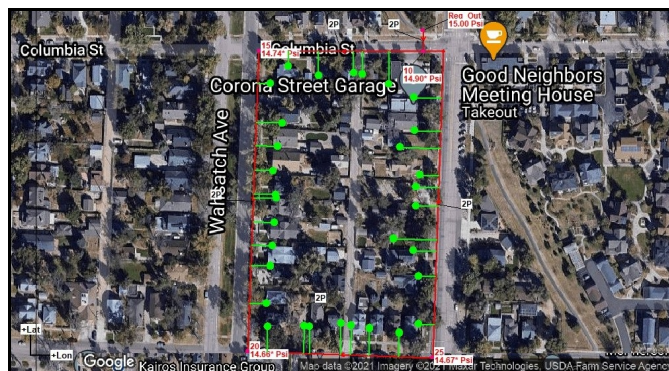
GDI Display

Directly above the GDI Display is the GDI Prompt Line. It is used to enter information needed by the various GDI Commands and to prompt the User for a required action. When a value is required, an entry field will be displayed for the GDI Prompt Line. Data is entered by typing in the provided entry field and pressing the *Enter* key. When a selection is required, a drop-down list will be displayed for the GDI Prompt Line. Select the desired item from the GDI Prompt List.

From Node Location:

GDI Prompt Line

Directly below the GDI Display are the Information Panel and the GDI Command Line. The Information Panel displays the coordinates for the current mouse position in the Information Panel. The coordinate values represent the current (scaled) position of the cursor. The Information Panel may also display the status of certain commands, such as whether the display of the customer symbols is "on" or "off." GDI Commands may be executed by typing their identifying abbreviation or alias on the GDI Command Line.

X (Lon) = -104.817698 Y (Lat) = 38.852359 | Zoom Level = 17

GDI Information Line (Top) / GDI Command Line (Bottom)

Along the right side of the GDI Display there are several "display tabs" - GDI Panels (*Item B*). Each GDI Panel can contain a different view of the model. Select the associated *Panel* tab to change to the desired view. Different graphic settings may be set for each GDI Panel. However, the Facility Settings are common for all of the GDI Panels.

Panel 1

Panel 2

Panel 3

Item B

The area to the right of the GDI Display, is known as the Data Panel. It serves several functions, allowing access to various settings and actions, the favorite results list, displaying data values and results, and allowing data values to be edited for selected features.

Settings & Actions...

[-] Settings...
   Set Data Panel Width
   Background Image Settings
   Color Display Settings
   Facility Settings
   Graphic Settings
   Text Display Settings
[-] Actions - Select One...
   Cancel Current GDI Command
   Undo Last Change
   Add An Action...
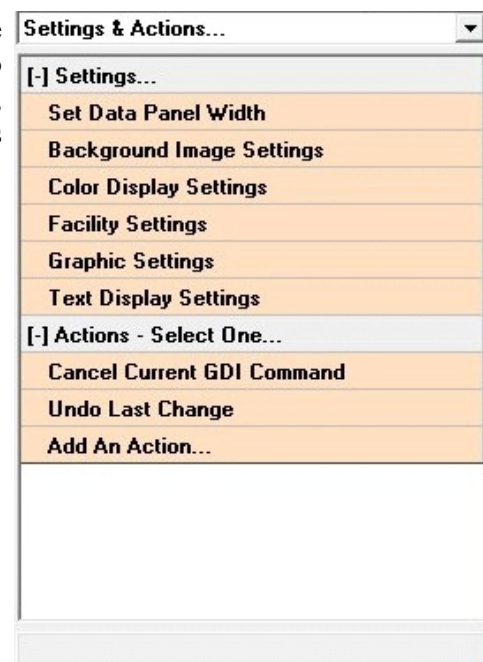
The area to the right of the GDI and below the Data Panel may display either the GDI Command List or the Tool Palette. Only one of the two may be displayed at the same time. Both of these items are described later in this section.
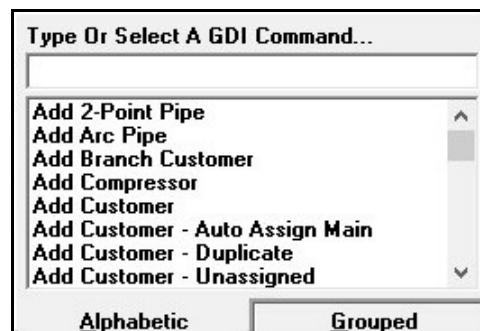
GDI Command List (Displayed) / Tool Palette

GDI Window Icons

Cancel - Cancels the currently executed GDI Command or stops the display of the model image from being repainted. The icon is located in the upper-left corner of the GDI.

GDI Command List - Toggles the display of the GDI Command List. When clicked, the GDI Command List is displayed in the lower-right corner of the GDI. Depending on the last tab used, either the *Alphabetic* list or the *Grouped* list will be displayed. When the icon is clicked again, the GDI Command List will be "hidden." The icon is located in the upper-right corner of the GDI.

GeoMap - Enables the GeoMap functions. Executing this command will display a message asking whether to continue with the GeoMap functions. When turned "on," the GeoMap functions allow the use of geographic map services to help in building or maintaining a model.

Feature Snap - Turns the Feature Snap "on" or "off" depending on the current status. The Feature Snap routine attempts to force (snap) new coordinate selections to the location of an existing feature location. When the Feature Snap is turned "on"and a coordinate entry prompt is displayed, a circle/target will be displayed about the center of the mouse crosshairs (Snap Target). The radius of the target will be the same as the Snap Target. When a new feature location is selected, GASWorkS will search for an existing feature within the Snap Target of the selected location. If a feature falls within the circle, its coordinates will be used for the prompted entry value. If no feature is found within the Snap Target, the selected location will be used. The Snap Target is set to the product of the *Feature Snap Ratio* value times the default *Node Symbol Size* value. To change either of these values, use the Graphic Settings screen. The icon is located in the lower-left corner of the GDI.

Quick Solve - Executes the Quick Solve routine using the currently set solution options. The icon is located in the lower-right corner of the GDI. The Solution Data screen will not be displayed; however, the solution results will be updated upon completion.
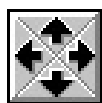
Scroll - Moves the model image up or down or right or left. Moving the image is called "panning" or "scrolling." The amount that the image moves when the *Scroll* icon is clicked is determined by the value of the *Pan/Scroll Change* values found on the Graphic Settings screen. To pan the GDI Image in a certain direction, click the arrow symbol corresponding to that direction on the *Scroll* icon. The icon is located in the lower-left corner of the GDI.

Solve - Displays the model's state, and when clicked, the Solution Data screen is displayed. The icon indicates whether the model is balanced (blue balance scale) or unbalanced (red unbalanced scale). When a change is made to the model data, the model is considered "unbalanced." The icon is located in the lower-right corner of the GDI.

Tool Palette - Toggles the display of the Tool Palette. When clicked, the Tool Palette is displayed in the lower-right corner of the GDI. The Tool Palette displays a limited number of User-selected GDI Commands. When the icon is clicked again, the Tool Palette will be "hidden." The icon is located in the upper-right corner of the GDI.

Zoom To Fit - Redisplays the entire model image to fit within the GDI Display. The icon is located in the lower-left corner of the GDI.

Zoom In - Enlarges (zooms in) the model image. The icon is located in the lower-left corner of the GDI.

Zoom Out - Reduces (zooms out) the model image. The icon is located in the lower-left corner of the GDI.

The GDI supports X-Y coordinate values ranging between negative and position 1 Billion, longitude between -180 and 180 degrees in longitude, and -90 and 90 degrees in latitude. The *Shift Model Coordinates* command may shift, scale, and rotate a model's coordinate values if they fall outside the allowable range. The *Coordinates* units are set on the Default Data Values screen. When a new pipe is added, its *Length* value is calculated based on the graphical length between its From Node, To Node, and vertex coordinates. The User may override the calculated value by manually entering the desired *Length* value. Node Names are automatically assigned during new pipe entry. The User may override the assigned node Name by manually typing the desired value. The *Node Increment* (difference between new node Names) is controlled by the value of the *Node Increment* item found on the *Miscellaneous* data tab on the Default Data Values screen.

GASWorkS™ 10.1 *GDI*

Depending on the zoom scale, certain features may or may not be displayed. The display of data text values is controlled by a User-specified setting on the Text Display Settings screen. The display of features like the flow arrows, pipe symbols, supply flag, and User Graphic Symbols are controlled by a User-specified setting on the Graphic Settings screen. Symbol sizes may be specified in one of two methods - absolute or relative. With the absolute method, the symbol size is expressed in terms of the *Coordinates* units. As the model image is enlarged or reduced, the display of the symbols is resized accordingly. With the relative method, a factor expressed as a percentage is used to specify the symbol size. The factor is applied to the diagonal distance across the GDI Display. The result is that the symbol size is always displayed at the same size regardless of the zoom scale. When an absolute size is used, and the image is zoomed to a scale where the symbols appear very small, they are not displayed. As the magnification of the GDI Image is increased, the symbols will eventually appear. When an absolute or relative size is used, the associated "display limit" determines whether the feature will be displayed at a particular zoom scale.

GDI Toolbars
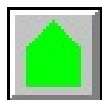
Customer Commands - Contains icons for commands used for manipulating the customer features.

Data Edit Commands - Contains icons for commands used for viewing and editing the data values for the pipe and node features.

Display Controls - Contains icons for commands used for controlling the display of the data text, feature colors, symbols, and the background image.
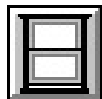
GDI Window Controls - Contains icons for commands used for controlling the GDI.
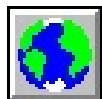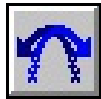
GeoMap Functions - Contains icons for commands used for controlling the GeoMap Functions.
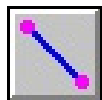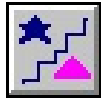
**Graphic Construction Commands -** Contains icons for commands used for adding and deleting pipe features.

Graphic Edit Commands - Contains icons for commands used for editing the pipe, vertex, and node features.
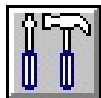
GASWorkS™ 10.1   *GDI*

Undo/Redo Commands - Contains icons for commands used for undoing (restoring) and redoing (restoring an undo) data changes.

User Graphic Commands - Contains icons for commands used for manipulating User Graphic Symbol, Line, and Image features.
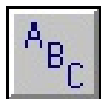
User Text Commands - Contains icons for commands used for manipulating User Text features.

Utility Commands - Contains icons for commands used for performing various utility commands such as plotting and tracing.

View Controls - Contains icons for commands used for zooming and panning the model image.

GDI Commands & Icon Descriptions

The GDI Commands may be executed in three manners - by selecting the GDI Command from the GDI Command List, by clicking on a GDI Icon from the various GDI Toolbars or the Tool Palette, or by typing the associated abbreviation or alias on the GDI Command Line. The methods are described in the following sections.

All of the GDI Commands may be accessed from the GDI Command List. Not all of the GDI Commands have a GDI Icon associated with them, and therefore, not all of the GDI Commands may be accessed by the GDI Toolbars or Tool Palette. Not all of the GDI Commands have an abbreviation associated with them, and therefore, not all of the GDI Commands may be accessed by the GDI Command Line.

GDI Command Line Method

The GDI Command Line allows the GDI Commands to be executed using an abbreviation or an alias for the GDI Command. The abbreviations are listed later in this section. Instructions on creating User-defined alias values may be found in the *Command Alias File* topic in the *Miscellaneous* section.
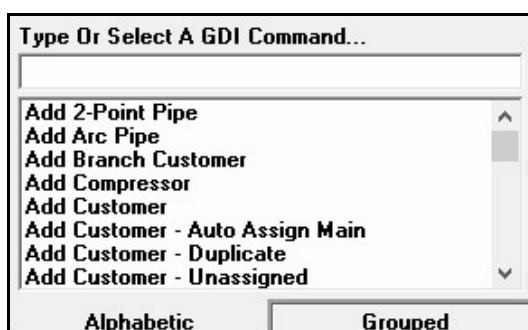
To execute a GDI Command from the GDI Command Line, type the abbreviation or alias in the GDI Command Line entry field, then press the *Enter* key.

GDI Commands may be entered in lower, upper, or proper case and can contain spaces in their syntax. For example, the *Add 2-Point Pipe* command may be entered as ADDPIPE, addpipe, or Add Pipe.
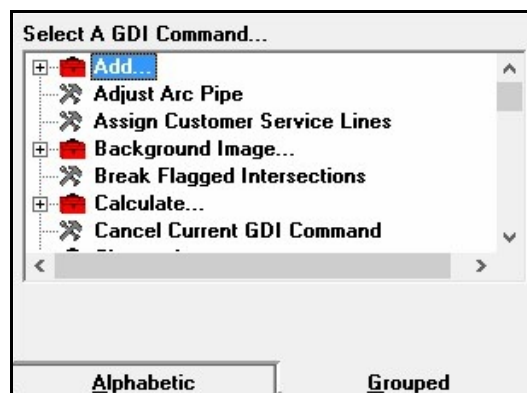

GDI Command List Method

Click the *GDI Command List* icon to display the GDI Command List in the lower-right corner of the GDI. The GDI Command List presents all of the GDI Commands in two different formats - one is an Alphabetic list, and the other is a Grouped list in a "treed" format.

When the *Alphabetic* list is displayed, begin typing the GDI Command in the data field. As the command is typed, the GDI Commands with matching characters will be displayed. Select the desired GDI Command to execute it.

When the *Grouped* list is displayed, click on a heading to display the associated GDI Commands, then click on the desired GDI Command to execute it.


GDI Toolbar Method

To select a GDI Command from a GDI Toolbar, move the cursor over the desired GDI Toolbar. The GDI Toolbar will be expanded. Click the desired GDI Icon.

GASWorkS™ 10.1

GDI

Tool Palette Method

Click the *Tool Palette* icon to display the Tool Palette in the lower-right corner of the GDI. The Tool Palette can display a limited number of User-selected GDI Commands.
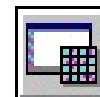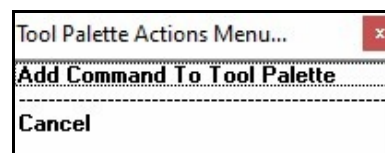
GDI Commands may be added to the Tool Palette from either the GDI Toolbar or the GDI Command List.
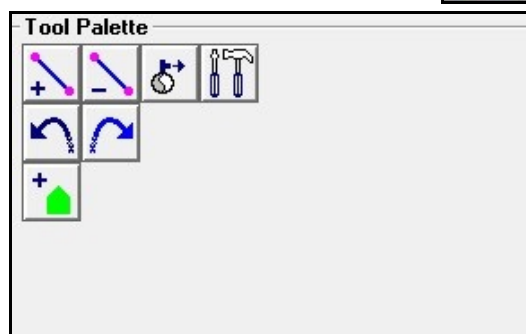
To add a command to the Tool Palette from a GDI Toolbar:

● Right-click on the desired icon.

● The Tool Palette Actions Menu will be displayed. Select the *Add Command To Tool Palette* item. The associated icon will be added to the Tool Palette.

To add a command to the Tool Palette from the GDI Command List:

● Left-click on a command to highlight the desired command, then right-click on the command.

● The Tool Palette Actions Menu will be displayed. Select the *Add Command To Tool Palette* item. The associated icon will be added to the Tool Palette. If the command has no associated icon, a generic icon will be displayed on the Tool Palette.

To select a GDI Command from the Tool Palette, double-click the desired icon. Icons on the Tool Palette may be rearranged by clicking and holding the left mouse button over the icon, then dragging it to a new location on the Tool Palette.

The Tool Palette can only hold as many icons as fit in the display window. If the Tool Palette is full, an icon must be removed before another icon may be added. To remove an icon from the Tool Palette, right-click on the desired icon. The Tool Palette Actions Menu will be displayed. Select the *Remove Icon From Tool Palette* item. The associated icon will be removed.

GASWorkS™ 10.1   *GDI*
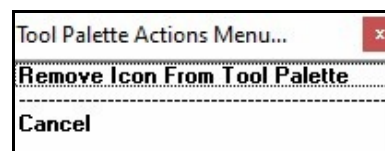
Additional Information

The Help Guide contains numerous examples and how-to tips for using the GDI Commands.

The various methods available for entering coordinates are described later in this section.

If the *Allow Data Entry During New Feature Entry* graphic settings option is selected, the following events will occur when a new pipe feature is added. If one or both of the associated nodes are new (does not already exist), the associated Node Data will be displayed in the Data Panel to allow entry of the new node data values. When both nodes have been entered, the Pipe Data will be displayed in the Data Panel to allow entry of the new pipe's data values.

GDI Command Descriptions

Add 2-Point Pipe - Adds a 2-point pipe feature (a straight pipe with only a From Node and To Node). Executing this command will cause a series of prompts to appear to identify the From Node and To Node locations.
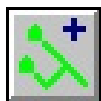> *Toolbar:* Graphic Construction Commands
> *GDI Command Line:* ADDPIPE, ADDP

Add Arc Pipe - Adds an arc-style pipe feature. Executing this command will cause a series of prompts to appear to identify the From Node and To Node locations and the pipe's curvature.
> *Toolbar:* Graphic Construction Commands
> *GDI Command Line:* ADDARCPIPE, ADDARC

Add Branch Customer - Adds a branch-style customer feature. A branch customer can only be added to an existing non-branch customer feature. Executing this command will cause a series of prompts to appear to identify the branch customer location, the "trunk" customer, and the tap location.
> *Toolbar:* Customer Commands
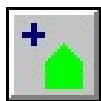> *GDI Command Line:* ADDBRANCHCUST, ADDBRANCH
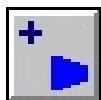
Add Compressor - Adds a compressor type "pipe" feature. Executing this command will cause a series of prompts to appear to identify the compressor's From Node and To Node locations. After the node locations have been selected, additional prompts will be displayed to complete the process. *Note* - When entering a compressor, the From Node must represent the upstream (suction) side of the compressor, and the To Node must represent the downstream (discharge) side of the compressor.
> *Toolbar:* Graphic Construction Commands
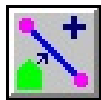> *GDI Command Line:* ADDCOMPRESSOR, ADDCOMP

Add Customer - Adds a single customer feature. Executing this command will cause a series of prompts to appear to identify the customer location and the supply main for the customer.
> *Toolbar:* Customer Commands
> *GDI Command Line:* ADDCUSTOMER, ADDCUST

Add Customer - Auto Assign Main - Adds a single customer feature and automatically assigns the customer to the main nearest the selected customer location. Executing this command will cause a prompt to appear to identify the customer location; then, the supply main/pipe will automatically be assigned.
> *Toolbar:* Customer Commands
> *GDI Command Line:* ADDAUTOCUSTOMER, AUTOCUST

Add Customer - Duplicate - Adds a new customer feature by duplicating an existing customer feature. Executing this command will cause a series of prompts to appear to identify the customer to be duplicated and the number of new customers to add. The new customer(s) will be placed at the same location (existing customer location) with the same data values.
> *Toolbar:* None
> *GDI Command Line:* ADDDUPLICATECUSTOMER, ADDDUP

Add Customer - Unassigned - Adds a customer feature but does not require a supply main assignment. Executing this command will cause a prompt to appear to identify the customer's location. *Note* -The customer may be assigned to a supply main at another time using the *Reassign Customer Supply Main* command.
> *Toolbar:* None
> *GDI Command Line:* ADDUNASSIGNED, ADDUNCUST

Add Customer - With Tap - Adds a customer feature and allows the User to select the tap location on the main. Executing this command will cause a series of prompts to appear to identify the customer location, the supply main/pipe, and the tap location for the customer.
> *Toolbar:* None
> *GDI Command Line:* ADDCUSTOMERTAP, ADDTAP

Add Feature To Group - Adds a pipe, User Graphic, or User Text feature to an existing group. Executing this command will cause a series of prompts to appear to identify the existing group to add the features to and to select the features to be added to the group. *Note* - A group must be created before features may be added to it.
> *Toolbar:* None
> *GDI Command Line:* ADDTOGROUP, ADDGRP
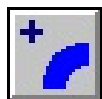
Add Fitting - Adds a fitting type "pipe" feature. Executing this command will cause a series of prompts to appear to identify the fittings From Node and To Node locations. After the node locations have been selected, additional prompts will be displayed to complete the process.
> *Toolbar:* Graphic Construction Commands
> *GDI Command Line:* ADDFITTING, ADDFIT

Add Multiple Customers - Adds multiple customer features. Executing this command will cause the *Add Customer* command to be repeated until the command is terminated by right-clicking the mouse, clicking the *Cancel* icon, or pressing the *Esc* key.
> *Toolbar:* None
> *GDI Command Line:* ADDMULTICUST

Add Multiple Customers - Auto Assign Main - Adds multiple customer features and automatically assigns the supply main/pipe. Executing this command will cause the *Add Customer - Auto Assign Main* command to be repeated until the command is terminated by right-clicking the mouse, clicking the *Cancel* icon, or pressing the *Esc* key.
> *Toolbar:* Customer Commands
> *GDI Command Line:* ADDMULTIAUTO

Add Polyline Pipe - Adds a polyline pipe feature (a pipe with a From Node, To Node, and deflections (vertices) in between). Executing this command will cause a series of prompts to appear to identify the From Node Location and the pipe vertex points. To end the vertex entry, right-click the mouse or press the *E* key.
> *Toolbar:* Graphic Construction Commands
> *GDI Command Line:* ADDPOLYLINEPIPE, ADDPOLY

Add Polyline Pipe Vertex - Adds a vertex to an existing polyline or 2-point pipe feature. Executing this command will cause a series of prompts to appear to identify the pipe to be modified and the location of the new vertex.
> *Toolbar:* Graphic Edit Commands
> *GDI Command Line:* ADDPIPEVERTEX, ADDPVTX

Add Polyline Service Customer - Adds a customer feature with a polyline style service line. Executing this command will cause a series of prompts to appear to identify the main tap location and the service line route (vertex points).
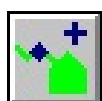> *Toolbar:* Customer Commands
> *GDI Command Line:* ADDPOLYSERVICE, ADDPOLYS
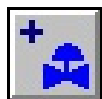
Add Regulator - Adds a regulator type "pipe" feature. Executing this command will cause a series of prompts to appear to identify the regulator's From Node and To Node locations. After both node locations have been selected, additional prompts will be displayed to complete the process. *Note -* When entering a regulator, the From Node must represent the upstream (high-pressure) side of the regulator, and the To Node must represent the downstream (low-pressure) side of the regulator.

> *Toolbar:* Graphic Construction Commands
> *GDI Command Line:* ADDREGULATOR, ADDREG

Add Service Attribute Marker - Adds a vertex to an existing customer service line and sets it as an attribute marker. Executing this command will cause a series of prompts to appear to select the customer to add the service vertex to and the new vertex location. This command is only available in the UKDM version of the GASWorkS software.

> *Toolbar:* Customer Commands
> *GDI Command Line:* ADDSATT, ADDSERVICEATT

Add Service Governor - Adds a governor to an existing customer service line. Executing this command will cause a series of prompts to appear to select the customer to add the service governor to, the governor's location, and then information about the governor. This command is only available in the UKDM version of the GASWorkS software.

> *Toolbar:* None
> *GDI Command Line:* ADDSERVICE

Add Service Line Vertex - Adds a vertex to an existing customer service line. Executing this command will cause a series of prompts to appear to identify the customer to be modified and the new vertex location.
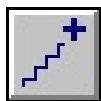
> *Toolbar:* Customer Commands
> *GDI Command Line:* ADDSERVICEVERTEX, ADDSVTX

Add User Graphic Image - Adds a User Graphic Image from a file. Executing this command will cause a series of prompts to appear to identify the image file to add, and the image location and width. *Note -* The contents of the *User Graphic Image Filename* list represent those files found in the "support\images" directory (folder) of the GASWorkS root directory (folder).
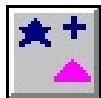
> *Toolbar:* User Graphic Commands
> *GDI Command Line:* ADDIMAGE, ADDI

GASWorkS™ 10.1   *GDI*

Add User Graphic Line - Adds a User Graphic Line. Executing this command will cause a series of prompts to appear to identify the line style, the width, and the Start and End Locations.
> *Toolbar:* User Graphic Commands
> *GDI Command Line:* ADDLINE, ADDL

Add User Graphic Symbol - Adds a User Graphic Symbol. Executing this command will cause a series of prompts to appear to identify the symbol style, location, rotation, and size.
> *Toolbar:* User Graphic Commands
> *GDI Command Line:* ADDSYMBOL, ADDS

Add User Text - Adds a User Text feature. Executing this command will cause a series of prompts to appear to identify the text location, rotation, height, width, and text value.
> *Toolbar:* User Text Commands
> *GDI Command Line:* ADDUSERTEXT, ADDT

Add User Text - Associated - Adds a User Text feature associated with a specific customer, node, or pipe feature's data text value. Executing this command will cause a series of prompts to appear to identify the feature to associate the User Text with, the data item to display, the text location, rotation, height, and whether a leader line will be included.
> *Toolbar:* None
> *GDI Command Line:* ADDAT

Add User Text - Predefined - Adds a User Text feature from an ASCII text file (.txt). Executing this command will cause a series of prompts to appear to identify the User Text location, rotation, height, width, and the ASCII text file to use. *Note* - The contents of the *Predefined User Text Filename* list are those files found in the "support\images" directory (folder) of the GASWorkS root directory (folder). This command is only available in the UKDM version of the GASWorkS software.
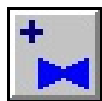> *Toolbar:* None
> *GDI Command Line:* ADDPREDEFINEDTEXT, ADDPT

Add User Text - With Leader Line - Adds a User Text feature with a leader line. Executing this command will cause a series of prompts to appear to identify the text location, rotation, height, text value, and the leader line information.
> *Toolbar:* None
> *GDI Command Line:* ADDLEADERTEXT, ADDLT

GASWorkS™ 10.1 *GDI*

Add Valve - Adds a valve type "pipe" feature. Executing this command will cause a series of prompts to appear to identify the valve's From Node and To Node locations. After both node locations have been selected, additional prompts will be displayed to complete the process.
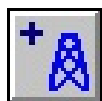   *Toolbar:* Graphic Construction Commands
    GDI Command Line: ADDVALVE, ADDV

Add Valve Node - Adds a valve node to an existing pipe feature. Executing this command will cause a series of prompts to appear to identify the pipe to add the valve node to, the location of the valve node, and the valve node information. This command is similar to the *Tap Pipe* command.
    Toolbar: None
    GDI Command Line: ADDVALVENODE, ADDVN

Add Well - Adds a well-type "pipe" feature. Executing this command will cause a series of prompts to appear to identify the wells From Node and To Node locations. After both node locations have been selected, additional prompts will be displayed to complete the process.
    Toolbar: Graphic Construction Commands
    GDI Command Line: ADDWELL

Adjust Arc Pipe - Adjusts the radius of an existing arc-style pipe feature. Executing this command will cause a series of prompts to appear to identify the pipe to be adjusted and the new radius.
    Toolbar: None
    GDI Command Line: ADJUSTARC

Assign Customer Service Lines - Automatically assigns the customer features to the nearest supply main/pipe. Executing this command will cause a message to be displayed asking whether to assign all of the customer service lines or only assign the service lines for the unassigned customers - respond appropriately. A prompt will be displayed to enter the maximum service length value. GASWorkS will process the selected customer types, find the main nearest the customer location, and assign the customer to that main. A 2-point "service line" will be placed from the customer symbol location to a perpendicular (where possible) tap location on the assigned pipe.
    Toolbar: None
    GDI Command Line: ASSIGNCUSTSERV, ASSIGNSERVICE

GASWorkS™ 10.1

Break Flagged Intersections - Automatically breaks the flagged unbroken intersection locations. Executing this command will cause a message to be displayed asking whether to really break all of the flagged locations - respond appropriately. If the *Yes* command button is clicked, the "header" pipe at each flagged intersection will be tapped, and the "lateral" pipe will be connected to the new tap. *Note -* Use the *Flag Unbroken Intersections* command to find and flag the unbroken intersections. *Note -* Use this feature with caution; it is strongly recommend that each flagged intersection location be reviewed before executing this command, or unusual pipe connections could result.

     Toolbar:
     GDI Command Line: BREAKFLAGGED

Calculate All Service Sizes - Calculates the service line sizes for all of the customer features in the model. Executing this command will cause the service line sizes to be calculated for all of the customers in the model. A message will be displayed when the calculation is complete. This command is only available in the UKDM version of the GASWorkS software.

     Toolbar: None - Icon in the lower-right corner of GDI Window
     GDI Command Line: CALCALLSIZES, CALCULATEALLSIZES

Calculate Delta P (Linear) - Calculates the pressure difference between a pair of selected nodes. Executing this command will cause a series of prompts to appear to identify the two nodes. A message will be displayed stating the calculated linear absolute pressure difference between the selected nodes.

     Toolbar: None
     GDI Command Line: CALCULATEDELTAP, CALCDP

Calculate Delta P (Squared) - Calculates the difference of the squared pressures between a pair of selected nodes. Executing this command will cause a series of prompts to appear to identify the two nodes. A message will be displayed stating the calculated difference in the squared absolute pressures between the selected nodes.

     Toolbar: None
     GDI Command Line: CALCULATEDELTAP2, CALCDP2

Calculate Design Factor - Calculates the Design Factor value. Executing this command will open the Design Factor Calculation screen. Enter and select the appropriate values, click the *Calculate* command button, then click the *Apply* command button.

     Toolbar: None
     GDI Command Line: CALCULATEDESIGNFACTOR, CALCDF

Calculate Efficiency - Calculates the hydraulic efficiency for the pipes along the flow path between a pair of selected nodes. Executing this command will cause a series of prompts to appear to identify the two nodes and the desired new *Pressure* value. An iterative series of calculations will compute the *Pipe Efficiency* value along the flow path between the selected nodes required to match the specified *Pressure* value. *Note -* There should only be *Pipe* Hydraulic Types features between the selected nodes, and all of the pipes should be assigned the same pipe *Flow Equation*.

>*Toolbar:* None
>*GDI Command Line:* CALCULATEEFFICIENCY, CALCEFF

Cancel Current GDI Command - Cancels the currently running GDI Command or stops the display of the model image if it is being repainted. Executing this command will cause the execution of a running GDI Command to be terminated. This command can interrupt the model image repaint (drawing) process before completion.

>*Toolbar:* None - Icon located near the upper-left corner of the GDI Window
>*GDI Command Line:* CANCEL

Clear All Customer Flags - Clears all customer flags (find and item) from the GDI Display. Executing this command will cause the display of the customer flags to be turned "off."

>*Toolbar:* None
>*GDI Command Line:* CLEARALLCUSTFLAG, CLEARALLCFLAG

Clear All Flags - Clears all feature flags (find, item, error, and supply) from the GDI Display. Executing this command will cause the display of the flags to be turned "off."

>*Toolbar:* None
>*GDI Command Line:* CLEARALLFLAGS, CLEARALL

Clear All Node Flags - Clears all node flags (find, item, and error) from the GDI Display. Executing this command will cause the display of all node flags to be turned "off."

>*Toolbar:* None
>*GDI Command Line:* CLEARALLNODEFLAG, CLEARALLNFLAG

Clear All Pipe Flags - Clears all pipe flags (find and item) from the GDI Display. Executing this command will cause the display of all pipe flags to be turned "off."
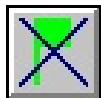
>*Toolbar:* None
>*GDI Command Line:* CLEARALLPIPEFLAG, CLEARALLPFLAG

GASWorkS™ 10.1 *GDI*

Clear Change Log - Clears (removes) the contents of the Change Log. Executing this command will cause the data in the Change Log to be cleared (deleted).
> *Toolbar:* None
> *GDI Command Line:* CLEARLOG

 Clear Customer Find Flag - Clears the customer "find" flag(s) from the GDI Display. Executing this command will cause the display of the flag identifying a "found" customer to be turned "off."
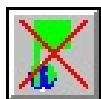> *Toolbar:* Customer Commands
> *GDI Command Line:* CLEARCUSTFINDFLAG, CLEARCFFLAG

Clear Customer Flag - Clears an individual (User-selected) customer flag from the GDI Display. Executing this command will cause a prompt to appear to identify the customer whose flag will be turned "off."
> *Toolbar:* None
> *GDI Command Line:* CLEARCUSTFLAG, CLEARCFLAG

Clear Customer Item Flags - Clears the customer "item" flags from the GDI Display. Executing this command will cause the display of all customer item flags to be turned "off."
> *Toolbar:* None
> *GDI Command Line:* CLEARCUSTITEMFLAG, CLEARCIFLAG

Clear DXF Find Flag - Clears the DXF text "find" flag from the GDI Display. Executing this command will cause the display of the flag identifying a "found" DXF text location to be turned "off."
> *Toolbar:* None
> *GDI Command Line:* CLEARDXFFLAG, CLEARDXF

Clear Error Flags - Clears the Data Check "error" flags from the GDI Display. Executing this command will cause the display of all error flags to be turned "off."
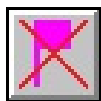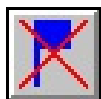> *Toolbar:* None
> *GDI Command Line:* CLEARERRORFLAG, CLEARERROR

 Clear GeoFind Flag - Clears the geographic "find" flag from the GDI Display. Executing this command will cause the display of the flag identifying a "found" geographic location to be turned "off."
> *Toolbar:* GeoMap Functions
> *GDI Command Line:* CLEARGEOFIND

GASWorkS™ 10.1 *GDI*

Clear Node Find Flag - Clears the node "find" flag from the GDI Display. Executing this command will cause the display of the flag identifying a "found" node to be turned "off."
> *Toolbar:* Data Edit Commands
> *GDI Command Line:* CLEARNODEFINDFLAG, CLEARNFFLAG

Clear Node Flag - Clears an individual (User-selected) node flag from the GDI Display. Executing this command will cause a prompt to appear to identify the node whose flag will be turned "off."
> *Toolbar:* None
> *GDI Command Line:* CLEARNODEFLAG, CLEARNFLAG

Clear Node Item Flags - Clears the node "item" flags from the GDI Display. Executing this command will cause the display of all node item flags to be turned "off."
> *Toolbar:* None
> *GDI Command Line:* CLEARNODEITEMFLAG, CLEARNIFLAG

Clear Pipe Find Flag - Clears the pipe "find" flag from the GDI Display. Executing this command will cause the display of the flag identifying a "found" pipe to be turned "off."
> *Toolbar:* Data Edit Commands
> *GDI Command Line:* CLEARPIPEFINDFLAG, CLEARPFFLAG

Clear Pipe Flag - Clears an individual (User-selected) pipe flag from the GDI Display. Executing this command will cause a prompt to appear to identify the pipe whose flag will be turned "off."
> *Toolbar:* None
> *GDI Command Line:* CLEARPIPEFLAG, CLEARPFLAG

Clear Pipe Item Flags - Clears the pipe "item" flags from the GDI Display. Executing this command will cause the display of the pipe item flags to be turned "off."
> *Toolbar:* None
> *GDI Command Line:* CLEARPIPEITEMFLAG, CLEARPIFLAG

Clear Supply Flags - Clears the "supply" flags from the GDI Display. Executing this command will cause the display of the flags identifying supply nodes to be turned "off."
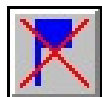> *Toolbar:* None
> *GDI Command Line:* CLEARSUPPLY, CLEARSFLAG

Clear Unbroken Intersection Flags - Clears the "unbroken" intersection flags from the GDI Display. Executing this command will cause the display of the flags identifying the unbroken intersections to be turned "off."

> *Toolbar:* None
> *GDI Command Line:* CLEARUNBROKEN

Clear User Text Find Flag - Clears the User Text "find" flag from the GDI Display. Executing this command will cause the display of the flag identifying a "found" User Text feature to be turned "off."

> *Toolbar:* User Text Commands
> *GDI Command Line:* CLEARTEXTFLAG, CLEARTFLAG

Close GDI Window - Closes the GDI (Graphic Data Interface) Window. Executing this command will cause the GDI to be closed. *Note -* The GDI cannot be closed when a GDI Command is running. To reopen the GDI, select the *View/Edit* item from the *Graphics* menu.

> *Toolbar:* GDI Window Controls
> *GDI Command Line:* CLOSE

Color Display Settings - Allows display colors to be set or changed. Executing this will cause the Color Display Settings screen to be displayed. Using the screen allows the default colors for model features to be set, creates a query to color features that meet specified criteria, and creates a color range based on a particular item value. Make the required changes, then click the *Close* command button.

> *Toolbar:* Display Controls
> GDI Command Line: SETCOLORS

Combine Pipes - Combines pipe features when the length of one of the pipe features is less than a User-specified length and only connected to one pipe. Executing this command will open the Pipe Combination Specifications screen. Make the required changes, then click the *Combine* command button. The model will be searched for pipes that may be combined using the User-specified requirements. When a node between two pipes of the same size is encountered, it will be deleted. *Note -* This command is useful when importing short pipes from a CAD or GIS system. This command is similar to the *Reduce Nodes* command.

> *Toolbar:* None
> *GDI Command Line:* COMBINEPIPES, COMBINE

Compare Two Models - Compares User-specified features and data values of the currently open model to a previously saved model. Executing this command will open the Model Selection screen. Select the desired model name to compare the currently open model, then click the *Continue* command button. The Model Comparison Report will be displayed stating the differences between the two models. Once the report is reviewed, click the *Close* command button.

> *Toolbar:* None
> *GDI Command Line:* COMPAREMODEL, COMPARE

Connect Pipe Ends - Connects a group of pipe features to a common node. Executing this command will cause a series of prompts to appear to identify the associated pipes whose ends need to be moved and the common node location.

> *Toolbar:* None
> *GDI Command Line:* CONNECTPIPEENDS, CONNECT

Convert All Arc Pipes To Polyline Pipes - Converts all of the arc pipe features to polyline pipe features. Executing this command will cause all arc pipes to be converted to polyline pipes.

> *Toolbar:* None
> *GDI Command Line:* CONVERTALLARCS, ALLARC2POLY

Convert Arc Pipe To Polyline Pipe - Converts an arc pipe feature to a polyline pipe feature. Executing this command will cause a prompt to appear to identify the arc pipe to be converted.

> *Toolbar:* None
> *GDI Command Line:* CONVERTARC, ARC2POLY

Convert Color Legend To User Text - Converts the Color Legend into a group of User Text features. Executing this command will cause a series of prompts to identify the text location, rotation, and the height. *Note -* The Color Legend must be converted to User Text to be plotted.

> *Toolbar:* None
> *GDI Command Line:* CONVERTCOLORLEGEND, CONVERTCL

Convert DXF Line To Pipe - Converts a line displayed in a DXF background image to a pipe feature. Executing this command will cause a persistent prompt to appear to identify the DXF line(s) to convert to pipe features. To end the selection process, right-click the mouse. The selected DXF lines will be converted to pipe features. *Note -* Only Arc, Line and Polyline type background entities may be converted to a pipe feature. This command is only available in the UKDM version of the GASWorkS software.

> *Toolbar:* None
> *GDI Command Line:* CONVERTDXF, DXF2PIPE

Convert DXF Line To Pipe And Offset - Creates a pipe feature from a line displayed in a DXF background image and allows the pipe location to be offset from the original line location. Executing this command will cause a persistent prompt to appear to identify the DXF line(s) to convert to pipes. To end the selection process, right-click the mouse. A prompt will be displayed to specify the offset distance and direction. *Note -* Only Arc, Line and Polyline type background entities may be converted to a pipe. This command is only available in the UKDM version of the GASWorkS software.

> *Toolbar:* None
> *GDI Command Line:* CONVERTDXFOFF, DXFOFFPIPE

Convert Service Lines To Customers - Converts a pipe feature assigned to a specified Facility Type to a customer feature and service line. Executing this command will cause a series of prompts to appear to identify the Facility Type to use and enter the *Fuzzy Tolerance* value. During the execution of this command, each "pipe" feature assigned to the selected Facility Type will be processed. If one end of the pipe falls within the Fuzzy Tolerance of a pipe assigned to another facility, the pipe will be converted to a service line. When converted, the pipe within the Fuzzy Tolerance will be assigned as the supply main/pipe; a customer feature will be created and placed at the end of the service line away from the supply main/pipe. *Note-* The associated Facility Type must be set to "Active" in the Facility Settings to convert the pipe features.

> *Toolbar:* None
> *GDI Command Line:* CONVERTSERVICE, SERVICE2CUST

Copy Customer - Copies an existing customer feature to one or more User-specified locations. Executing this command will cause a series of prompts to appear to identify the customer feature to be copied, the new customer location, and the supply main. A new customer is placed at the selected location with the same data values as the selected customer. This command will be repeated until the mouse is right-clicked or the *Cancel* icon is clicked.

> *Toolbar:* None
> *GDI Command Line:* COPYCUSTOMER, COPYC

Copy Customer & Service - Copies an existing customer feature and its service line to one or more User-specified locations. Executing this command will cause a series of prompts to appear to identify the customer feature to be copied and the new customer location. A new customer is placed at the selected location with the same supply main and data values as the selected customer. This command will be repeated until the mouse is right-clicked or the *Cancel* icon is clicked.

> *Toolbar:* None
> *GDI Command Line:* COPYCUSTOMERSERVICE, COPYCS

GASWorkS™ 10.1   *GDI*

Copy Group - Copies a previously created group to one or more User-specified locations. Executing this command will cause a series of prompts to appear to identify the group to copy, the "base point" for the group, and the new location. A copy of the "base" group will be placed at the selected location. This command will be repeated until the mouse is right-clicked or the *Cancel* icon is clicked.
> *Toolbar:* None
> *GDI Command Line:* COPYGROUP, COPYG

Copy Pipe - Copies an existing pipe feature to a User-specified location. Executing this command will cause a series of prompts to appear to identify the pipe to be copied, the "base" node, and the new location. A new pipe with the same data values as the selected "base" pipe will be placed at the selected location.
> *Toolbar:* None
> *GDI Command Line:* COPYPIPE, COPYP

Copy Pipe - Offset - Copies an existing pipe feature and offsets it at a User-specified value. Executing this command will cause a series of prompts to appear to identify the pipe to be copied, the offset distance, and the offset side of the original pipe. A new pipe will be placed at the offset location with the same data values as the selected "base" pipe. This command is only available in the UKDM version of the GASWorkS software.
> *Toolbar:* None
> *GDI Command Line:* OFFPIPE, OFFSETPIPE

Copy Saved Views From Model - Copies all saved views from a previously saved model to the currently open model. Executing this command will open the Model Selection screen. Select the desired model name with the views to copy, then click the *Continue* command button. *Note* - The views associated with the currently open model will be replaced by the views saved in the selected model.
> *Toolbar:* None
> *GDI Command Line:* COPYVIEWS, COPYV

Copy User Graphic Image - Copies an existing User Graphic Image to one or more User-specified locations. Executing this command will cause a series of prompts to appear to identify the feature to be copied, a "base point" for the image, and the new location. A copy of the "base" feature will be placed at the selected location. This command will be repeated until the *Esc* key is pressed, the mouse is right-clicked, or the *Cancel* icon is clicked.
> *Toolbar:* None
> *GDI Command Line:* COPYIMAGE, COPYI

Copy User Graphic Line - Copies an existing User Graphic Line to one or more User-specified locations. Executing this command will cause a series of prompts to appear to identify the feature to be copied, the "base point" for the line, and the new location. A copy of the "base" feature will be placed at the selected location. This command will be repeated until the *Esc* key is pressed, the mouse is right-clicked, or the *Cancel* icon is clicked.

> *Toolbar:* None
> *GDI Command Line:* COPYLINE, COPYL

Copy User Graphic Symbol - Copies an existing User Graphic Symbol to one or more User-specified locations. Executing this command will cause a series of prompts to appear to identify the feature to be copied, the "base point" for the symbol, and the new location. A copy of the "base" feature will be placed at the selected location. This command will be repeated until the *Esc* key is pressed, the mouse is right-clicked, or the *Cancel* icon is clicked.

> *Toolbar:* None
> *GDI Command Line:* COPYSYMBOL, COPYS

Copy User Graphics From Model - Copies all of the User Graphic features (Lines and Symbols) from a previously saved model to the currently open model. Executing this command will open the Model Selection screen. Select the desired model name with the User Graphics to copy, then click the *Continue* command button. *Note* - The User Graphics from the selected model will be appended (added) to the graphics associated with the currently open model.

> *Toolbar:* None
> *GDI Command Line:* COPYALLGRAPHICS

Copy User Table - Copies an existing User Table to one or more User-specified locations. Executing this command will cause a series of prompts to appear to identify the User Table to be copied, the "base point" for the table, and the new location. A new User Table with the same characteristics as the selected "base" User Table will be placed at the selected location. This command will be repeated until the *Esc* key is pressed, the mouse is right-clicked, or the *Cancel* icon is clicked.

> *Toolbar:* None
> *GDI Command Line:* COPYTABLE

Copy User Text - Copies an existing User Text feature to one or more User-specified locations. Executing this command will cause a series of prompts to appear to identify the feature to be copied and the new location. A new User Text feature with the same characteristics as the selected "base" feature will be placed at the selected location. This command will be repeated until the *Esc* key is pressed, the mouse is right-clicked, or the *Cancel* icon is clicked.

> *Toolbar:* None
> *GDI Command Line:* COPYTEXT, COPYT

Copy User Text From Model - Copies all of the User Text features from a previously saved model to the currently open model. Executing this command will open the Model Selection screen. Select the desired model name with the User Text features to copy, then click the *Continue* command button.

> *Toolbar:* None
> *GDI Command Line:* COPYALLTEXT

Create Bill Of Materials - Creates a Bill Of Materials for the pipe features, fittings, valves, tees, and governors in a model. Executing this command will cause the Bill Of Materials table to be displayed. This command is only available in the UKDM version of the GASWorkS software.

> *Toolbar:* None
> *GDI Command Line:* CREATEBOM, BOM

Create Branch Customer Tap - Creates a branch and trunk style customer feature from two existing customer features. Executing this command will cause a series of prompts to appear to identify the branch customer, the trunk customer, and the new tap location.

> *Toolbar:* Customer Commands
> *GDI Command Line:* CREATEBRANCHCUST, CREATEBRANCH

Create Group - Creates a group of pipe, User Graphics, and/or User Text features. Executing this command will cause a series of prompts to appear to identify the features to add to the group and the Group Type.

> *Toolbar:* None
> *GDI Command Line:* CREATEGROUP

Create Symbol Legend - Creates a legend (in table form) of the User Graphic Symbols that are used in the model. Executing this command will cause a series of prompts to appear to identify the Symbol Legend location, the table width and height, the User Graphic Symbol size, and the number of columns in the table.

> *Toolbar:* None
> *GDI Command Line:* CREATESYMBOLLEGEND, CREATESYMLEG

Create User Table - Creates and adds a User Table. Executing this command will cause a series of prompts to appear to enter the table title and the number of rows and columns. The User Table Editor screen is displayed. Enter the desired information in the various data cells, then click the *Close* command button. A prompt will be displayed to identify the table location and enter the text height.

> *Toolbar:* None
> *GDI Command Line:* CREATETABLE

Delete All Customers - Deletes all customer features in the current model. Executing this command will cause a message to be displayed asking whether to really delete all of the customers - respond appropriately.
> *Toolbar:* None
> *GDI Command Line:* DELETEALLCUST

Delete All Marked Views - Deletes all marked views in the current model. Executing this command will cause a message to be displayed asking whether to really delete all of the marked views - respond appropriately.
> *Toolbar:* None
> *GDI Command Line:* DELETEALLVIEWS

Delete All User Graphics - Deletes all User Graphic (Image, Line, and Symbol) features in the current model. Executing this command will cause a message to be displayed asking whether to really delete all of the User Graphics - respond appropriately.
> *Toolbar:* None
> *GDI Command Line:* DELETEALLGRAPHICS

Delete All User Text - Deletes all User Text features in the current model. Executing this command will cause a message to be displayed asking whether to really delete all of the User Text features - respond appropriately.
> *Toolbar:* None
> *GDI Command Line:* DELETEALLTEXT

Delete All Valve Nodes - Deletes all valve nodes in the current model. Executing this command will cause a message to be displayed asking whether to really delete all of the valve nodes- respond appropriately.
> *Toolbar:* None
> *GDI Command Line:* DELETEALLVALVENODES, DELVNODES

Delete Bill Of Materials Table - Deletes the Bill Of Materials Table. Executing this command will cause a message to be displayed asking whether to really delete the Bill Of Materials Table - respond appropriately. This command is only available in the UKDM version of the GASWorkS software.
> *Toolbar:* None
> *GDI Command Line:* DELETEBOMTABLE, DELBOMT

Delete Color Legend Text - Deletes the Color Legend Text. Executing this command will cause a message to be displayed asking whether to really delete the Color Legend Text- respond appropriately.
> *Toolbar:* None
> *GDI Command Line:* DELETECOLOR

GASWorkS™ 10.1   *GDI*
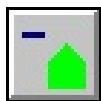
Delete Customer - Deletes a customer feature. Executing this command will cause a prompt to appear to identify the customer to be deleted.
> *Toolbar:* Customer Commands
> *GDI Command Line:* DELETECUSTOMER, DELC

Delete Group - Deletes an existing group. Executing this command will cause a prompt to appear to identify a feature in the group to be deleted. A message will be displayed asking whether to really delete the features in the selected group - respond appropriately.
> *Toolbar:* None
> *GDI Command Line:* DELETEGROUP, DELG

Delete Marked View - Deletes a marked view. Executing this command will cause a prompt to appear to identify the marked view to be deleted.
> *Toolbar:* None
> *GDI Command Line:* DELETEMARKEDVIEW, DELMVIEW

Delete Node - Deletes a node feature. Executing this command will cause a prompt to appear to identify the node to be deleted. This command is similar to the *Join Adjacent Pipes* command.
> *Toolbar:* None
> *GDI Command Line:* DELETENODE, DELN

Delete Pipe - Deletes a pipe feature. Executing this command will cause a prompt to appear to identify the pipe to be deleted.
> *Toolbar:* Graphic Construction Commands
> *GDI Command Line:* DELETEPIPE, DELP

Delete Polyline Pipe Vertex - Deletes a vertex from a polyline pipe feature. Executing this command will cause a series of prompts to appear to identify the pipe to delete the vertex from and the vertex to be deleted.
> *Toolbar:* Graphic Edit Commands
> *GDI Command Line:* DELETEPOLYVERTEX, DELPVTX
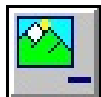
Delete Saved View - Deletes a previously saved view. Executing this command will cause a prompt to appear to identify the name of the view to be deleted.
> *Toolbar:* View Controls
> *GDI Command Line:* DELETESAVEDVIEW, DELSVIEW

Delete Service Governor - Deletes an existing service governor from a customer service line. Executing this command will cause a prompt to appear to identify the customer to delete the governor. This command is only available in the UKDM version of the GASWorkS software.

　　Toolbar: None
　　GDI Command Line: DELETESERVICE, DELSERV

Delete Service Line Vertex - Deletes a vertex from a polyline-style service line. Executing this command will cause a series of prompts to appear to identify the customer to delete the vertex from and the vertex to be deleted.

　　Toolbar: Customer Commands
　　GDI Command Line: DELETESERVICEVERTEX, DELSVTX

Delete User Graphic Image - Deletes a User Graphic Image. Executing this command will cause a prompt to appear to identify the User Graphic Image to be deleted.

　　Toolbar: User Graphic Commands
　　GDI Command Line: DELETEIMAGE, DELI
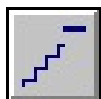
Delete User Graphic Line - Deletes a User Graphic Line. Executing this command will cause a prompt to appear to identify the User Graphic Line to be deleted.

　　Toolbar: User Graphic Commands
　　GDI Command Line: DELETELINE, DELL
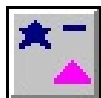
Delete User Graphic Symbol - Deletes a User Graphic Symbol. Executing this command will cause a prompt to appear to identify the User Graphic Symbol to be deleted.

　　Toolbar: User Graphic Commands
　　GDI Command Line: DELETESYMBOL, DELS

Delete User Table - Deletes a User Table. Executing this command will cause a prompt to appear to identify the User Table to be deleted.

　　Toolbar: None
　　GDI Command Line: DELETETABLE, DELTAB

Delete User Text - Deletes a User Text feature. Executing this command will cause a prompt to appear to identify the User Text feature to be deleted.

　　Toolbar: User Text Commands
　　GDI Command Line: DELETETEXT, DELT

Delete Valve Node - Deletes a valve node. Executing this command will cause a prompt to appear to identify the valve node to be deleted. This command is similar to the *Join Adjacent Pipes* command.
> *Toolbar:* None
> *GDI Command Line:* DELETEVALVENODE, DELVN

Delete Zero Length Pipes - Deletes all pipe features with a zero (0) *Length* value. Executing this command will cause a message to be displayed asking whether to really delete all zero-length pipes - respond appropriately.
> *Toolbar:* None
> *GDI Command Line:* DELETEZEROPIPES
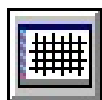
Display Background Image - Turns the display of all background images "on" or "off" depending on the current status.
> *Toolbar:* Display Controls
> *GDI Command Line:* DISPLAYBACKGROUND, DISPBACK

Display Color Legend - Turns the display of the Color Legend "on" or "off" depending on the current status. *Note -* The Color Legend will remain displayed until turned "off." During GDI Commands, the Color Legend will be hidden so that the entire GDI is visible. Move the cursor over the *Display Color Legend* icon to redisplay the hidden Color Legend.
> *Toolbar:* Display Controls
> *GDI Command Line:* DISPLAYCOLOR, DISPCOLOR
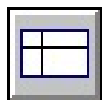
Display Crosshairs - Turns the display of the mouse cursor crosshairs "on" or "off" depending on the current status.
> *Toolbar:* GDI Window Controls
> *GDI Command Line:* DISPLAYCROSSHAIRS, CROSSHAIRS

Display Customer Symbols - Turns the display of the customer symbols "on" or "off" depending on the current status. Depending on the zoom scale of the model image and the *Customer Symbol Size*, the symbols may or may not be visible.
> *Toolbar:* Customer Commands
> *GDI Command Line:* DISPLAYCUSTOMER, DISPCUST

GASWorkS™ 10.1 *GDI*

Display Flow Arrows - Turns the display of the flow direction arrows "on" or "off" depending on the current status. Depending on the zoom scale of the model image and *Flow Arrow Size*, the arrows may or may not be visible.
>> *Toolbar:* Display Controls
>> *GDI Command Line:* DISPLAYFLOW, FLOWARROW

Display Isometric View - Turns the Isometric (3D) View "on" and the Plan View "off." To turn "off" the Isometric View, use the *Display Plan View* command.
>> *Toolbar:* None
>> *GDI Command Line:* ISOMETRICVIEW, ISO

Display Node Symbols - Turns the display of the node symbols "on" or "off" depending on the current status. Depending on the zoom scale of the model image and the node symbol size, the symbols may or may not be visible.
>> *Toolbar:* Display Controls
>> *GDI Command Line:* DISPLAYNODE, DISPNODE

Display Pipe Symbols - Turns the display of the pipe symbols "on" or "off" depending on the current status. Depending on the zoom scale of the model image and *Pipe Symbol Size*, the symbols may or may not be visible. Pipe symbols are displayed for non-pipe hydraulic features, including compressor, regulator, valve, and well-type features.
>> *Toolbar:* Display Controls
>> *GDI Command Line:* DISPLAYPIPE, DISPPIPE

Display Plan View - Turns the Plan View "on" and the Isometric (3D) View "off." To turn "off" the Plan View, use the *Display Isometric View* command.
>> *Toolbar:* None
>> *GDI Command Line:* PLANVIEW, PLAN
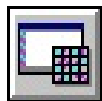
Display Tool Palette - Displays the Tool Palette in the lower-right corner of the GDI.
>> *Toolbar:* None - Icon located near the upper-right corner of the GDI Window
>> *GDI Command Line:* DISPLAYTOOL, TOOLPALETTE

Display User Graphics - Turns the display of the User Graphics (Images, Lines, and Symbols) "on" or "off" depending on the current status.
>> *Toolbar:* User Graphic Commands
>> *GDI Command Line:* DISPLAYGRAPHICS

GASWorkS™ 10.1　　　　　　　　　　　*GDI*

Display User Text - Turns the display of the User Text features "on" or "off" depending on the current status.
> *Toolbar:* User Text Commands
> *GDI Command Line:* DISPLAYTEXT

Dissolve All Groups - Dissolves (ungroups) all existing groups in the current model. Executing this command will cause a message to be displayed asking whether to dissolve all of the groups in the model - respond appropriately.
> *Toolbar:* None
> *GDI Command Line:* DISSOLVEALL, UNGROUPALL

Dissolve Group - Dissolves (ungroups) all of its features from an existing group. Executing this command will cause a prompt to appear to select a feature in the desired group to dissolve. A message will be displayed asking whether to really dissolve the group features - respond appropriately.
> *Toolbar:* None
> *GDI Command Line:* DISSOLVEGROUP, UNGROUP

DXF Layer Identification - Identifies the layer that a selected DXF entity is on. Executing this command will cause a prompt to appear to identify the DXF entity to be selected. Select a DXF entity, and a message will be displayed listing the layer information about the selected DXF entity.
> *Toolbar:* None
> *GDI Command Line:* DXFLAYER

Edit Customer Data - Displays the data values associated with a customer feature. Executing this command will cause a prompt to appear to identify the customer to edit. The data associated with the selected feature will be displayed in the Data Panel. If changes were made, click the *Apply Data Values* command button.
> *Toolbar:* Customer Commands
> *GDI Command Line:* EDITCUSTOMER, EDITCUST

Edit Customer Data By Pipe Selection - Displays the data values of the customer features assigned to a selected pipe feature. Executing this command will cause a prompt to appear to identify the desired pipe with the attached customer features to be edited. Once selected, the Multiple Customers Found screen is displayed. Select a Customer ID Number to display the associated customer data in the Data Panel.
> *Toolbar:* None
> *GDI Command Line:* EDITCUSTBYPIPE

Edit Multiple Customers - Allows changes to be made to a selected group of customer features. Executing this command will cause a prompt to appear to identify the customer features to be edited. Once selected, the Multiple Edit Specifications screen is displayed. Make the required changes or execute the desired command.
> *Toolbar:* Customer Commands
> *GDI Command Line:* EDITMULTICUST
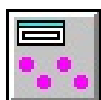
Edit Multiple Nodes - Allows changes to be made to a selected group of nodes. Executing this command will cause a prompt to appear to identify the nodes to be edited. Once selected, the Multiple Edit Specifications screen is displayed. Make the required changes or execute the desired command.
> *Toolbar:* Data Edit Commands
> *GDI Command Line:* EDITMULTINODE
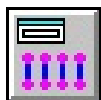
Edit Multiple Pipes - Allows changes to be made to a selected group of pipe features. Executing this command will cause a prompt to appear to identify the pipe features to be edited. Once selected, the Multiple Edit Specifications screen is displayed. Make the required changes or execute the desired command.
> *Toolbar:* Data Edit Commands
> *GDI Command Line:* EDITMULTIPIPE

Edit Multiple User Graphic Images - Allows changes to be made to a selected group of User Graphic Images. Executing this command will cause a prompt to appear to identify the features to be edited. Once selected, the Multiple Edit Specifications screen is displayed. Make the required changes or execute the desired command.
> *Toolbar:* None
> *GDI Command Line:* EDITMULTIUGIMAGE, EDITMULTIUGI

Edit Multiple User Graphic Lines - Allows changes to be made to a selected group of User Graphic Lines. Executing this command will cause a prompt to appear to identify the features to be edited. Once selected, the Multiple Edit Specifications screen is displayed. Make the required changes or execute the desired command.
> *Toolbar:* None
> *GDI Command Line:* EDITMULTIUGLINE, EDITMULTIUGL

Edit Multiple User Graphic Symbols - Allows changes to be made to a selected group of User Graphic Symbols. Executing this command will cause a prompt to appear to identify the features to be edited. Once selected, the Multiple Edit Specifications screen is displayed. Make the required changes or execute the desired command.
> *Toolbar:* None
> *GDI Command Line:* EDITMULTIUGSYMBOL, EDITMULTIUGS

Edit Multiple User Text - Allows changes to be made to a selected group of User Text features. Executing this command will cause a prompt to appear to identify the features to be edited. Once selected, the Multiple Edit Specifications screen is displayed. Make the required changes or execute the desired command.
> *Toolbar:* None
> *GDI Command Line:* EDITMULTIUSERTEXT, EDITMULTIUT

 Edit Node Data - Displays the data values associated with a node. Executing this command will cause a prompt to appear to identify the node to edit. The data associated with the selected feature will be displayed in the Data Panel. If changes are made, click the *Apply Data Values* command button.
> *Toolbar:* Data Edit Commands
> *GDI Command Line:* EDITNODE

Edit Node External Load - Changes the external load calculated value for a selected node. Executing this command will cause a prompt to appear to identify the node to be modified and the new *External Load* value. *Note* - Use of this command may corrupt the automatic external load accounting. Particular care should be taken when using this command.
> *Toolbar:* None
> *GDI Command Line:* EDITNODELOAD

Edit Node External Load (Fixed) - Changes the "fixed" external load calculated value for a selected node. Executing this command will cause a prompt to appear to identify the node to edit and the new *External Load (Fixed)* value. *Note* - Use of this command may corrupt the automatic external load accounting. Particular care should be taken when using this command.
> *Toolbar:* None
> *GDI Command Line:* EDITNODELOADFIX

Edit Pipe Customer Count - Changes the *Customer Count* value for a selected pipe feature. Executing this command will cause a series of prompts to appear to identify the pipe to edit and the new *Customer Count* value. *Note -* Use of this command may corrupt the automatic external load accounting. Particular care should be taken when using this command.
> *Toolbar:* None
> *GDI Command Line:* EDITPIPECOUNT
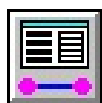
Edit Pipe Data - Displays the data values associated with a pipe feature. Executing this command will cause a prompt to appear to identify the pipe to edit. The data associated with the selected feature will be displayed in the Data Panel. If changes are made, click the *Apply Data Values* command button.
> *Toolbar:* Data Edit Commands
> *GDI Command Line:* EDITPIPE

Edit Polyline Pipe Vertex - Displays the data values associated with a polyline pipe vertex. Executing this command will cause a prompt to appear to identify the polyline pipe vertex to edit. The data associated with the selected feature will be displayed in the Data Panel. If changes are made, click the *Apply Data Values* command button.
> *Toolbar:* None
> *GDI Command Line:* EDITPOLYVERTEX, EDITPVTX

Edit User Graphic Image - Displays the data values associated with a User Graphic Image. Executing this command will cause a prompt to appear to identify the feature to edit. The data associated with the selected feature will be displayed in the Data Panel. If changes are made, click the *Apply Data Values* command button.
> *Toolbar:* User Graphic Commands
> *GDI Command Line:* EDITIMAGE, EDITI
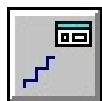
Edit User Graphic Line - Displays the data values associated with a User Graphic Line. Executing this command will cause a prompt to appear to identify the feature to edit. The data associated with the selected feature will be displayed in the Data Panel. If changes are made, click the *Apply Data Values* command button.
> *Toolbar:* User Graphic Commands
> *GDI Command Line:* EDITLINE, EDITL

GASWorkS™ 10.1   *GDI*

Edit User Graphic Symbol - Displays the data values associated with a User Graphic Symbol. Executing this command will cause a prompt to appear to identify the feature to edit. The data associated with the selected feature will be displayed in the Data Panel. If changes are made, click the *Apply Data Values* command button.
> *Toolbar:* User Graphic Commands
> *GDI Command Line:* EDITSYMBOL, EDITS

Edit User Table - Displays the data values associated with a User Table. Executing this command will cause a prompt to appear to identify a text or graphic feature associated with the User Table to edit. The User Table Editor screen is displayed. Click in the desired cell and type the desired value. When finished, click the *Close* command button.
> *Toolbar:* None
> *GDI Command Line:* EDITTABLE

Edit User Text - Displays the data values associated with a User Text feature. Executing this command will cause a prompt to appear to identify the feature to edit. The data associated with the selected feature will be displayed in the Data Panel. If changes are made, click the *Apply Data Values* command button.
> *Toolbar:* User Text Commands
> *GDI Command Line:* EDITTEXT, EDITT

Export User Text - Exports the User Text features to a text file (.txt). Executing this command will open the File Selection screen. Enter or select the desired name, then click the *Continue* command button.
> *Toolbar:* None
> *GDI Command Line:* EXPORTTEXT

Feature Snap - Turns the Feature Snap "on" or "off" depending on the current status. The Feature Snap attempts to force (snap) new coordinate selections to an existing feature within the Snap Target of the selected location. When the Feature Snap is turned "on" and a coordinate entry prompt is displayed, a "target" will be displayed about the center of the mouse crosshairs (Snap Target). The radius of the target will be the same as the Snap Target. When a new feature location is selected, GASWorkS will search for an existing feature within the Snap Target of the selected location. If a feature falls within the target, its coordinates will be used for the prompted entry value. If no feature is found within the Snap Target, the selected location will be used. The size of the Snap Target is set to the product of the *Feature Snap Ratio* value times the default *Node Symbol Size* value. To change either of these values, use the Graphic Settings screen. The icon is located in the lower-left corner of the GDI.
> *Toolbar:* None - Lower-left corner of the GDI Window
> *GDI Command Line:* SNAP

Find Any Feature - Finds and flags the location of a general feature (customer, DXF Text, node, pipe, or User Text) based on a User input value. Executing this command will cause a series of prompts to appear to identify what feature to find. If a match is found, a flag will be placed at the associated location. The flag is persistent and will remain displayed until turned "off" using the *Clear All Flags* command. *Note -* If an alphanumeric search item is selected, the case and syntax must be identical for a match. *Note -* This command may also be executed using the *CTRL+F* key combination.

> *Toolbar:* None
> *GDI Command Line:* FIND

Find Customer - Finds and flags a customer feature based on a User input value. Executing this command will cause a series of prompts to appear to identify the method to search by and the search value. If a match is found, a flag will be placed at the associated location. The flag is persistent and will remain displayed until turned "off" using the *Clear Customer Find Flag* command. *Note -* If an alphanumeric search item is selected and the term "LIKE" is entered before the value to be found, partial matches and similar case occurrences may be found. Otherwise, the case and syntax must be identical for a match.

> *Toolbar:* Customer Commands
> *GDI Command Line:* FINDCUSTOMER, FINDCUST

Find Customer By Address - Finds and flags a customer feature based on a User input *Street Address* value. Executing this command will cause a prompt to appear to identify the *Street Address* of the customer. If a match is found, a flag will be placed at the associated location. The flag is persistent and will remain displayed until turned "off" using the *Clear Customer Find Flag* command. *Note -* This command is only available when an attribute file (database) is attached and the *Street Address* field is assigned. If an alphanumeric search item is selected and the term "LIKE" is entered before the search value, partial matches, and similar case occurrences may be found. Otherwise, the case and syntax must be identical for a match. *Note -* A customer attribute file must be attached, and its status must be set to "on" for this command to be enabled.

> *Toolbar:* None
> *GDI Command Line:* FINDCUSTOMERADDRESS, FINDCUSTADD

Find Customer By Attribute - Finds and flags a customer feature based on a User input "attribute" value. Executing this command will cause a prompt to appear to identify the *Search String* value. If a match is found, a flag will be placed at the associated location. The flag is persistent and will remain displayed until turned "off" using the *Clear Customer Find Flag* command. *Note -* This command is only available when an attribute file is attached. If an alphanumeric search item is selected, the case and syntax must be identical for a match. *Note -* A customer attribute file must be attached, and its status must be set to "on" for this command to be enabled.

> *Toolbar:* None
> *GDI Command Line:* FINDCUSTOMERATTRIBUTE, FINDCUSTATT

GASWorkS™ 10.1   *GDI*

Find Customer By Internal ID Number - Finds and flags a customer feature by its Internal ID Number. Executing this command will cause a prompt to appear to identify the Internal ID Number of the customer. If a match is found, a flag will be placed at the associated location. The flag is persistent and will remain displayed until turned "off" using the *Clear Customer Find Flag* command.

> *Toolbar:* None
> *GDI Command Line:* FINDCUSTOMERID, FINDCID

Find Customer By Link ID Number - Finds and flags a customer feature by its Link ID Number. Executing this command will cause a prompt to appear to identify the Link ID Number of the customer. If a match is found, a flag will be placed at the associated location. The flag is persistent and will remain displayed until turned "off" using the *Clear Customer Find Flag* command.

> *Toolbar:* None
> *GDI Command Line:* FINDCUSTOMERLINKID, FINDCLINK

Find Customer By Record Number - Finds and flags a customer feature by its Record Number. Executing this command will cause a prompt to appear to identify the Record Number of the customer to find. If a match is found, a flag will be placed at the associated location. The flag is persistent and will remain displayed until turned "off" using the *Clear Customer Find Flag* command.

> *Toolbar:* None
> *GDI Command Line:* FINDCUSTOMERRECORD, FINDCREC

Find DXF Text - Finds and flags a DXF Text location based on a User input value. Executing this command will cause a prompt to appear to identify the DXF Text value to find. If a match is found, a flag will be placed at the associated location. The flag is persistent and will remain displayed until turned "off" using the *Clear DXF Find Flag* command. *Note -* If the term "LIKE" is entered before the value to be found, partial matches and similar case occurrences may be found. Otherwise, the case and syntax must be identical for a match.

> *Toolbar:* None
> *GDI Command Line:* FINDDXFTEXT, FINDDXF

 Find Geographic Location - Finds and flags a geographic location based on a User input street *Address* or geographic *Location Name*. Executing this command will cause a prompt to appear to identify the *Address* or *Location Name* value to find. If a match is found, a flag will be placed at the associated location. The flag is persistent and will remain displayed until turned "off" using the *Clear GeoFind Flag* command. *Note -* The GeoMap Functions must be enabled to use the command.

> *Toolbar:* GeoMap Functions
> *GDI Command Line:* GEOFIND

GASWorkS™ 10.1   *GDI*

Find Node - Finds and flags a node based on a User input value. Executing this command will cause a series of prompts to appear to identify the method to search by and the search value. If a match is found, a flag will be placed at the associated location. The flag is persistent and will remain displayed until turned "off" using the *Clear Node Find Flag* command. *Note -* If an alphanumeric search item is selected and the term "LIKE" is entered before the value to be found, partial matches and similar case occurrences may be found. Otherwise, the case and syntax must be identical for a match.

        Toolbar: Data Edit Commands
        GDI Command Line: FINDNODE

Find Node By Name - Finds and flags a node by its Name. Executing this command will cause a prompt to appear to identify the name of the node to find. If a match is found, a flag will be placed at the associated location. The flag is persistent and will remain displayed until turned "off" using the *Clear Node Find Flag* command. *Note -* If the term "LIKE" is entered before the value to be found, partial matches and similar case occurrences may be found. Otherwise, the case and syntax must be identical for a match.

        Toolbar: None
        GDI Command Line: FINDNODENAME, FINDNNAME

Find Node By Record Number - Finds and flags a node by its Record Number. Executing this command will cause a prompt to appear to identify the Record Number for the node to find. If a match is found, a flag will be placed at the associated location. The flag is persistent and will remain displayed until it is turned "off" using the *Clear Node Find Flag* command.

        Toolbar: None
        GDI Command Line: FINDNODERECORD, FINDNREC

Find Pipe - Finds and flags a pipe feature based on a User input value. Executing this command will cause a series of prompts to appear to identify the method to search by and the search value. If a match is found, a flag will be placed at the associated location. The flag is persistent and will remain displayed until turned "off" using the *Clear Pipe Find Flag* command. *Note -* If an alphanumeric search item is selected and the term "LIKE" is entered before the value to be found, partial matches and similar case occurrences may be found. Otherwise, the case and syntax must be identical for a match.

        Toolbar: Data Edit Commands
        GDI Command Line: FINDPIPE

GASWorkS™ 10.1   *GDI*

Find Pipe By Address - Finds and flags a pipe feature based on a User input *Address* value. Executing this command will cause a prompt to appear to identify the *Address* of the pipe to find. If a match is found, a flag will be placed at the associated location. The flag is persistent and will remain displayed until turned "off" using the *Clear Pipe Find Flag* command. *Note* - This command is only available when an attribute file (database) is attached and the *Address* field is assigned. If the term "LIKE" is entered before the value to be found, partial matches and similar case occurrences may be found. Otherwise, the case and syntax must be identical for a match.

>*Toolbar:* None
>*GDI Command Line:* FINDPIPEADDRESS, FINDPADD

Find Pipe By Attribute - Finds and flags a pipe feature based on a User input "attribute" data item value. Executing this command will cause a prompt to appear to enter the *Search String* value. If a match is found, a flag will be placed at the associated location. The flag is persistent and will remain displayed until turned "off" using the *Clear Pipe Find Flag* command. *Note* - This command is only available when an attribute file (database) is attached and the associated field is assigned. If an alphanumeric search item is selected and the term "LIKE" is entered before the value to be found, partial matches and similar case occurrences may be found. Otherwise, the case and syntax must be identical for a match.

>*Toolbar:* None
>*GDI Command Line:* FINDPIPEATTRIBUTE, FINDPATT

Find Pipe By Internal ID Number - Finds and flags a pipe feature by its Internal ID Number. Executing this command will cause a prompt to appear to identify the Internal ID Number of the pipe to find. If a match is found, a flag will be placed at the associated location. The flag is persistent and will remain displayed until turned "off" using the *Clear Pipe Find Flag* command.

>*Toolbar:* None
>*GDI Command Line:* FINDPIPEID, FINDPID

Find Pipe By Link ID Number - Finds and flags a pipe feature by its Link ID Number. Executing this command will cause a prompt to appear to identify the Link ID Number of the pipe to find. If a match is found, a flag will be placed at the associated location. The flag is persistent and will remain displayed until turned "off" using the *Clear Pipe Find Flag* command.

>*Toolbar:* None
>*GDI Command Line:* FINDPIPELINKID, FINDPLINK

Find Pipe By Record Number - Finds and flags a pipe feature by its Record Number. Executing this command will cause a prompt to appear to identify the Record Number of the pipe to find. If a match is found, a flag will be placed at the associated location. The flag is persistent and will remain displayed until turned "off" using the *Clear Pipe Find Flag* command.

>*Toolbar:* None
>*GDI Command Line:* FINDPIPERECORD, FINDPREC

Find User Text - Finds and flags a User Text feature based on a User input value. Executing this command will cause a prompt to appear to identify the User Text value. If a match is found, a flag will be placed at the associated location. The flag is persistent and will remain displayed until turned "off" using the *Clear User Text Find Flag* command. *Note -* If the term "LIKE" is entered before the value to be found, partial matches and similar case occurrences may be found. Otherwise, the case and syntax must be identical for a match.

> *Toolbar:* User Text Commands
> *GDI Command Line:* FINDTEXT, FINDT

Flag Customer - Finds and flags a customer feature based on a User input value. Executing this command will cause a prompt to appear to identify the customer to flag. A flag will be placed at the associated location. The flag is persistent and remain displayed until turned "off" using the *Clear Customer Flag* command. *Note - *The customer symbols must be turned "on" to display the flag locations.

> *Toolbar:* None
> *GDI Command Line:* FLAGCUST, FLAGC

Flag Customers By Item Value - Finds and flags customer features based on a User-specified item value. Executing this command will open the Query Specifications screen. Make the desired selection, then click the *Perform Query* command button. If a match is found, a flag will be placed at the associated location. Multiple matches may be found. The flags are persistent and will remain displayed until turned "off" using the *Clear Customer Item Flags* command. *Note -* The customer symbols must be turned "on" to display the flag locations.

> *Toolbar:* None
> *GDI Command Line:* FLAGCUSTOMERITEM, FLAGCUSTITEM

Flag Customers - Invalid Link ID Numbers - Finds and flags customer features with "invalid" Link ID Numbers. Executing this command will cause each customer whose Link ID Number is not found in the assigned customer attribute file to be marked with a flag. A message will be displayed stating the results of the routine. The Customer Link ID Number Report will be displayed. The flags are persistent and will remain displayed until turned "off" using the *Clear All Customer Flags* command. *Note -* The customer symbols must be turned "on" to display the flag locations.

> *Toolbar:* None
> *GDI Command Line:* FLAGBADCID

Flag Customers - Redundant Link ID Numbers - Finds and flags customer features with redundant Link ID Numbers. Executing this command will cause each customer whose Link ID Number is found multiple times in the assigned customer attribute file to be marked with a flag. A message will be displayed stating the results of the routine. The Customer Link ID Number Report will be displayed. The flags are persistent and will remain displayed until turned "off" using the *Clear All Customer Flags* command. *Note* - The customer symbols must be turned "on" to display the flag locations.

> *Toolbar:* None
> *GDI Command Line:* FLAG2XCID

Flag Customers - Unassigned - Finds and flags customer features not assigned to a supply main. Executing this command will cause GASWorkS to search for customers that are not assigned to a pipe or main. A message will be displayed stating the results of the routine. If a match is found, a flag will be placed at the associated location. Multiple matches may be found. The flags are persistent and will remain displayed until turned "off" using the *Clear Customer Find Flags* command. *Note* - The customer symbols must be turned "on" to display the flag locations.

> *Toolbar:* None
> *GDI Command Line:* FLAGUNASSIGNED, FLAGUNCUST

Flag Node - Finds and flags a User-specified node. Executing this command will cause a prompt to appear to identify the node to flag. A flag will be placed at the associated location. The flag is persistent and will remain displayed until turned "off" using the *Clear Node Flag* command. *Note* - The node symbols must be turned "on" to display the flag locations.

> *Toolbar:* None
> *GDI Command Line:* FLAGNODE, FLAGN

Flag Nodes By Item Value - Finds and flags nodes based on a User-specified item value. Executing this command will open the Query Specifications screen. Make the desired selection, then click the *Perform Query* command button. If a match is found, a flag will be placed at the associated location. Multiple matches may be found. The flag is persistent and will remain displayed until turned "off" using the *Clear Node Item Flags* command.

> *Toolbar:* None
> *GDI Command Line:* FLAGNODEITEM, FLAGNITEM

Flag Pipe - Finds and flags a User-specified pipe feature. Executing this command will cause a prompt to appear to identify the pipe to flag. A flag will be placed at the associated location. The flag is persistent and will remain displayed until turned "off" using the *Clear All Pipe Flags* command. *Note* - The pipe symbols must be turned "on" to display the flag locations.

> *Toolbar:* None
> *GDI Command Line:* FLAGPIPE, FLAGP

Flag Pipes By Item Value - Finds and flags pipe features based on a User-specified item value. Executing this command will open the Query Specifications screen. Make the desired selection, then click the *Perform Query* command button. If a match is found, a flag will be placed at the associated location. Multiple matches may be found. The flag is persistent and will remain displayed until turned "off" using the *Clear Pipe Item Flags* command.

> *Toolbar:* None
> *GDI Command Line:* FLAGPIPEITEM, FLAGPITEM

Flag Supply Nodes - Finds and flags nodes with positive load values (supply nodes). Executing this command will cause GASWorkS to search for nodes where the calculated *Total Load* value is positive (greater than zero). A flag will placed at each location where a supply node is found. The flags are persistent and remain displayed until turned "off" using the *Clear Supply Flags* command. *Note* - The node symbols must be turned "on" to display the flag locations.

> *Toolbar:* None
> *GDI Command Line:* FLAGSUPPLY

Flag Unbroken Intersections - Finds and flags nodes where a pipe, not connected to the node, is found within a specified tolerance. This often occurs when a lateral pipe is not connected to a header pipe. Executing this command will open the Fuzzy Tolerance screen. Enter the appropriate value, then click the *Continue* command button. GASWorkS will search for instances where a pipe end is located within the specified *Fuzzy Tolerance* value but not connected to it. A flag will be placed at each location where this is found. In some cases, this may not occur at an "intersection." The flags are persistent and will remain displayed until turned "off" using the *Clear Unbroken Intersection Flags* command. *Note* - The node symbols must be turned "on" to display the flag locations.

> *Toolbar:* None
> *GDI Command Line:* FLAGUNBROKEN

Get Group ID - Displays the Group ID Number for a User-selected group. Executing this command will cause a prompt to appear to identify a feature in the associated group. Once selected, a message will be displayed stating the Group ID Number.

> *Toolbar:* None
> *GDI Command Line:* GETGROUPID, GETGRP

Go To XY Location - Positions the upper-left corner of the model image at a User-specified XY-Coordinate location. Executing this command will cause a prompt to appear to identify the XY-Coordinates to move to.

> *Toolbar:* View Controls
> *GDI Command Line:* GOTOXY, GOTO

Highlight DXF Entity - Highlights a User-selected DXF entity and displays information about the selected entity. Executing this command will cause a prompt to appear to identify the DXF entity to be selected. Select an entity, and a message will be displayed listing information about the selected entity. *Note -* Highlighting a DXF entity is useful to identify which layer an entity is on or when trying to determine an entity's graphical type. This command is only available in the UKDM version of the GASWorkS software.
> *Toolbar:* None
> *GDI Command Line:* HIGHLIGHTDXF, HILITDXF

Highlight Group - Highlights the features in a User-selected group. Executing this command will cause a prompt to appear to identify an item in the desired group to highlight. The features will remain highlighted until the *Unhighlight Group* command is used.
> *Toolbar:* None
> *GDI Command Line:* HIGHLIGHTGROUP, HILITGRP

Highlight Pipe - Highlights a User-selected pipe feature, and displays the pipe Facility Type and Size Type/Code. Executing this command will cause a prompt to appear to identify the pipe to highlight.
> *Toolbar:* None
> *GDI Command Line:* HIGHLIGHTPIPE, HILITP

Insert Compressor - Inserts a compressor-type feature into an existing pipe-type feature. Executing this command will cause a series of prompts to appear to identify the pipe to insert the compressor into, the location of the compressor endpoints, and the operating data. *Note -* One must be at an existing node location, and one end must be at a new (tapped) location along the pipe the compressor is inserted into. The From Node must represent the upstream (suction) side of the compressor, and the To Node must represent the downstream (discharge) side of the compressor.
> *Toolbar:* None
> *GDI Command Line:* INSERTCOMPRESSOR, INCOMP

Insert Regulator - Inserts a regulator-type feature into an existing pipe-type feature. Executing this command will cause a series of prompts to appear to identify the pipe to insert the regulator into, the location of the regulator endpoints, and the operating data. *Note -* One end must be at an existing node location, and one end must be at a new (tapped) location along the pipe that the regulator is being inserted into. The From Node must represent the upstream (high-pressure) side of the regulator, and the To Node must represent the downstream (low-pressure) side of the regulator.
> *Toolbar:* None
> *GDI Command Line:* INSERTREGULATOR, INREG

Insert Valve - Inserts a valve-type feature into an existing pipe-type feature. Executing this command will cause a series of prompts to appear to identify the pipe to insert the valve into, the location of the valve endpoints, and the operating data. *Note -* One end must be at an existing node location, and one end must be at a new (tapped) location along the pipe that the valve is being inserted into. The From Node and To Node order do not matter for valve-type features.

> *Toolbar:* None
> *GDI Command Line:* INSERTVALVE, INV

Join Adjacent Pipes - Joins two adjacent (connected) pipe features. Executing this command will cause a series of prompts to appear to identify the pipes to be joined. During the Join routine, the node common to both pipes will be removed. The new (combined) pipe will possess the hydraulic and graphic data associated with the first pipe that was selected. *Note -* Arc pipe features cannot be joined. If one of the pipes to be joined is an arc pipe feature, a message will be displayed asking whether to convert it to a polyline pipe feature. Once converted, the selected pipe may be joined.

> *Toolbar:* Graphic Edit Commands
> *GDI Command Line:* JOINPIPE

Mark Current View - Marks the view currently displayed in the GDI Image with a cross-hatched area.

> *Toolbar:* None
> *GDI Command Line:* MARKVIEW

Match Customer Attribute Properties - Assigns the attribute data values from one customer feature to another customer feature. Executing this command will cause a series of prompts to appear to identify the "base" customer and the customer(s) to assign the base values. This command is only available in the UKDM version of the GASWorkS software.

> *Toolbar:* None
> *GDI Command Line:* MATCHCUSTOMERATTRIBUTE, MATCHCATT

Match Customer Graphic Properties - Assigns the graphic data values from one customer feature to one or more customer features. Executing this command will cause a series of prompts to appear to identify the "base" customer and the customer(s) to assign the base data values.

> *Toolbar:* None
> *GDI Command Line:* MATCHCUSTOMERGRAPHIC, MATCHCG

Match Customer Hydraulic Properties - Assigns the hydraulic data values from one customer feature to one or more customer features. Executing this command will cause a series of prompts to appear to identify the "base" customer and the customer(s) to assign the base data values.

> *Toolbar:* None
> *GDI Command Line:* MATCHCUSTOMERHYDRAULIC, MATCHCHYD

Match Customer Properties (All) - Assigns the graphic and hydraulic data values from one customer feature to one or more customer features. Executing this command will cause a series of prompts to appear to identify the "base" customer and the customer(s) to assign the base data values.

>　*Toolbar:* None
>　*GDI Command Line:* MATCHCUSTOMERPROP, MATCHC

Match Node Graphic Properties - Assigns the graphic data values from one node to one or more nodes. Executing this command will cause a series of prompts to appear to identify the "base" node and the node(s) to assign the base data values.

>　*Toolbar:* None
>　*GDI Command Line:* MATCHNODEGRAPHIC, MATCHNG

Match Node Hydraulic Properties - Assigns the hydraulic data values from one node to one or more nodes. Executing this command will cause a series of prompts to appear to identify the "base" node and the node(s) to assign the base data values.

>　*Toolbar:* None
>　*GDI Command Line:* MATCHNODEHYDRAULIC, MATCHNHYD

Match Node Properties (All) - Assigns both the graphic and hydraulic data values from one node to one or more nodes. Executing this command will cause a series of prompts to appear to identify the "base" node and the node(s) to assign the base data values.

>　*Toolbar:* None
>　*GDI Command Line:* MATCHNODEPROP, MATCHN

Match Pipe Attribute Properties - Assigns the attribute data values from one pipe feature to another pipe feature. Executing this command will cause a series of prompts to appear to identify the "base" pipe and the pipe(s) to assign the base data values. This command is only available in the UKDM version of the GASWorkS software.

>　*Toolbar:* None
>　*GDI Command Line:* MATCHPIPEATTRIBUTE, MATCHPATT

Match Pipe Facility - Assigns the *Facility Type* value from one pipe feature to one or more pipe-type features. Executing this command will cause a series of prompts to appear to identify the "base" pipe and the pipe(s) to assign the base *Facility Type* value.

>　*Toolbar:* None
>　*GDI Command Line:* MATCHPIPEFACILITY, MATCHPFAC

Match Pipe Graphic Properties - Assigns the graphic data values from one pipe feature to one or more pipe features. Executing this command will cause a series of prompts to appear to identify the "base" pipe and the pipe(s) to assign the base data values.
> *Toolbar:* None
> *GDI Command Line:* MATCHPIPEGRAPHIC, MATCHPG

Match Pipe Hydraulic Properties - Assigns the hydraulic data values from one pipe feature to one or more pipe features. Executing this command will cause a series of prompts to appear to identify the "base" pipe and the pipe(s) to assign the base data values.
> *Toolbar:* None
> *GDI Command Line:* MATCHPIPEHYDRAULIC, MATCHPHYD

Match Pipe Properties (All) - Assigns the graphic and hydraulic data values from one pipe feature to one or more pipe features. Executing this command will cause a series of prompts to appear to identify the "base" pipe and the pipe(s) to assign the base data values.
> *Toolbar:* None
> *GDI Command Line:* MATCHPIPEPROP, MATCHP

Match User Graphic Line Properties - Assigns the graphic data values from one User Graphic Line to one or more User Graphic Lines. Executing this command will cause a series of prompts to appear to identify the "base" line and the line(s) to assign the base data values.
> *Toolbar:* None
> *GDI Command Line:* MATCHLINEPROP, MATCHL

Match User Graphic Symbol Properties - Assigns the graphic data values from one User Graphic Symbol to one or more User Graphic Symbols. Executing this command will cause a series of prompts to appear to identify the "base" symbol and the symbol(s) to assign the base data values.
> *Toolbar:* None
> *GDI Command Line:* MATCHSYMBOLPROP, MATCHS

Match User Text Properties - Assigns the graphic data items from one User Text feature to one or more User Text features. Executing this command will cause a series of prompts to appear to identify the "base" text and the text item(s) to assign the base data values.
> *Toolbar:* None
> *GDI Command Line:* MATCHTEXT, MATCHT

GASWorkS™ 10.1 *GDI*
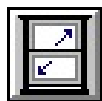
Maximize GDI Window - Increases the size of the Graphic Data Interface (GDI) Window to fill the entire GASWorkS display area.
> *Toolbar:* GDI Window Controls
> *GDI Command Line:* MAXIMIZEWINDOW, MAXWIN

Measure Angle - Calculates an angle based on two User-specified points. Executing this command will cause a series of prompts to appear to identify the two points to measure the angle of with respect to the GDI Coordinate System. Once selected, a message will be displayed stating the angle and rotation values. This command is only available in Plan View.
> *Toolbar:* None
> *GDI Command Line:* MEASUREANGLE, ANGLE
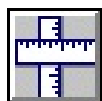
Measure Distance - Calculates a distance based on two or more User-specified points. Executing this command will cause a series of prompts to appear to identify the points to measure between. Right-click the mouse or press the *E* key to end the selection process. A message will be displayed stating the measured distances. This command is only available in Plan View.
> *Toolbar:* Utility Commands
> *GDI Command Line:* MEASUREDISTANCE, DISTANCE

Measure X Distance - Calculates a distance along the x-axis based on two User-specified points. Executing this command will cause a series of prompts to appear to identify the points to measure between. Once selected, a message will be displayed stating the x-axis distance.
> *Toolbar:* None
> *GDI Command Line:* MEASUREX, DISTANCEX

Measure Y Distance - Calculates a distance along the y-axis based on two User-specified points. Executing this command will cause a series of prompts to appear to identify the points to measure between. Once selected, a message will be displayed stating the y-axis distance.
> *Toolbar:* None
> *GDI Command Line:* MEASUREY, DISTANCEY

Measure Z Distance - Calculates a distance along the z-axis based on two User-specified points. Executing this command will cause a series of prompts to appear to identify the points to measure between. Once selected, a message will be displayed stating the z-axis distance. This command is only available in Isometric View.
> *Toolbar:* None
> *GDI Command Line:* MEASUREZ, DISTANCEZ

GASWorkS™ 10.1   *GDI*

Move Background Image - Moves an attached background image. Executing this command will cause a series of prompts to appear to identify the background image file to be moved, a "base point" for the image, and the new location.
> *Toolbar:* None
> *GDI Command Line:* MOVEBACKGROUNDIMAGE, MOVEBIMG

Move Bill Of Materials Table - Moves the Bill Of Materials Table. Executing this command will cause a series of prompts to appear to identify the "base point" for the Bill Of Materials Table and the new location. The Bill Of Materials Table will be placed at the selected location. This command is only available in the UKDM version of the GASWorkS software.
> *Toolbar:* None
> *GDI Command Line:* MOVEBOM

Move Branch Service Tap - Moves the service tap location associated with a branch customer feature from one trunk customer feature to another customer feature. Executing this command will cause a series of prompts to appear to identify the affected customers and the new service tap location.
> *Toolbar:* Customer Commands
> *GDI Command Line:* MOVEBRANCHTAP, MOVEBRANCH

Move Color Legend Text - Moves the Color Legend Text. Executing this command causes a prompt to identify a "base point" for the Color Legend Text and the new location.
> *Toolbar:* None
> *GDI Command Line:* MOVECOLOR
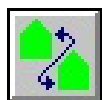
Move Customer - Moves a customer feature. Executing this command will cause a series of prompts to appear to identify the customer to be moved and the new location. *Note* - This command does not change the supply main assignment. *Note* - The customer "location" grip may also be used to move the customer location.
> *Toolbar:* Customer Commands
> *GDI Command Line:* MOVECUSTOMER, MOVEC

Move Customer - Reassign Main - Moves a customer feature and reassigns the supply main. Executing this command will cause a series of prompts to appear to identify the customer to be moved, the new location, and the new supply main. *Note* - The customer service "tap" grip may also reassign the supply main.
> *Toolbar:* None
> *GDI Command Line:* MOVECUSTREASSIGN

GASWorkS™ 10.1 *GDI*
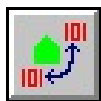
Move Customer Text - Moves the text item associated with a customer feature. Executing this command will cause a series of prompts to appear to identify the customer to move the text for, a "base point" for the text, and the new text location and rotation. *Note -* The customer "text" grip may also move the customer text location.

    Toolbar: Customer Commands
    GDI Command Line: MOVECUSTOMERTEXT, MOVECTEXT

Move Group - Moves the features in a group. Executing this command will cause a series of prompts to appear to identify the group to be moved, a "base point" for the group, and the new location.

    Toolbar: None
    GDI Command Line: MOVEGROUP, MOVEG

Move Node - Moves a node and its connected pipe features. Executing this command will cause a series of prompts to appear to identify the node to be moved and the new location. *Note -* The node "location" grip may also move the node location.

    Toolbar: Graphic Edit Commands
    GDI Command Line: MOVENODE, MOVEN
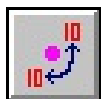
Move Node Text - Moves the text item associated with a node. Executing this command will cause a series of prompts to appear to identify the node to move the text for, a "base point" for the text, and the new location. *Note -* The node "text" grip may also move the node text location.

    Toolbar: Data Edit Commands
    GDI Command Line: MOVENODETEXT, MOVENTEXT

Move Node Text - Multiple - Moves the text items associated with multiple nodes. Executing this command will cause the *Move Node Text* command to be repeated until the *Esc* key is pressed, the mouse is right-clicked, or the *Cancel* icon is clicked.

    Toolbar: None
    GDI Command Line: MOVEMULTINODETEXT, MXNT

Move Pipe - Moves a pipe feature. Executing this command will cause a series of prompts to appear to identify the pipe to be moved, a "base point" for the pipe, and the new location.

    Toolbar: None
    GDI Command Line: MOVEPIPE, MOVEP
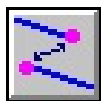
Move Pipe End - Moves a pipe feature end from one node to another. Executing this command will cause a series of prompts to appear to identify the associated pipe, the end to be moved, and the new location. *Note -*This command may connect or disconnect a pipe from a node. The pipe "node" grips can also move the pipe end location.
> *Toolbar:* Graphic Edit Commands
> *GDI Command Line:* MOVEPIPEEND, MOVEPEND

Move Pipe Text - Moves the text items associated with a pipe feature. Executing this command will cause a series of prompts to appear to identify the pipe to move the text for, a "base point" for the text, and the new text location and rotation. *Note -* The pipe "text" grip may also move the pipe text location.
> *Toolbar:* Data Edit Commands
> *GDI Command Line:* MOVEPIPETEXT, MOVEPTEXT

Move Pipe Text - Multiple - Moves the text items associated with multiple pipe features. Executing this command will cause the *Move Pipe Text* command to be repeated until the *Esc* key is pressed, the mouse is right-clicked, or the *Cancel* icon is clicked.
> *Toolbar:* None
> *GDI Command Line:* MOVEMULTIPIPETEXT, MXPT

Move Polyline Pipe Vertex - Moves a vertex for a polyline pipe feature. Executing this command will cause a prompt to appear to identify the polyline pipe vertex to be moved and the new location. *Note -* The pipe "vertex" grips can also move a vertex location.
> *Toolbar:* Graphic Edit Commands
> *GDI Command Line:* MOVEPOLYVERTEX, MOVEPVTX

Move Service Line Vertex - Moves a vertex for a polyline type customer service. Executing this command will cause a series of prompts to appear to identify the associated customer, the service line vertex to be moved, and the new location.
> *Toolbar:* Customer Commands
> *GDI Command Line:* MOVESERVICEVERTEX, MOVESVTX
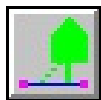
Move Service Tap - Moves a customer service line tap location. Executing this command will cause a series of prompts to appear to identify the associated customer and the new service tap location. *Note -* The customer service "tap" grip may also be used to move the tap location.
> *Toolbar:* Customer Commands
> *GDI Command Line:* MOVESERVICETAP, MOVESTAP

Move User Graphic Image - Moves a User Graphic Image. Executing this command will cause a series of prompts to appear to identify the User Graphic Image to be moved, a "base point" for the image, and the new location.
  Toolbar: User Graphic Commands
  GDI Command Line: MOVEIMAGE, MOVEI
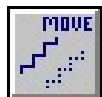
Move User Graphic Line - Moves a User Graphic Line. Executing this command will cause a series of prompts to appear to identify the User Graphic Line to be moved, a "base point" for the line, and the new location.
  Toolbar: User Graphic Commands
  GDI Command Line: MOVELINE, MOVEL

Move User Graphic Line End - Moves a User Graphic Line end or vertex. Executing this command will cause a series of prompts to appear to identify the User Graphic Line segment to be moved, the end to be moved, and the new location. *Note -* The graphic line "end" grips can also move a graphic line end.
  Toolbar: User Graphic Commands
  GDI Command Line: MOVELINEEND, MOVELE

Move User Graphic Symbol - Moves a User Graphic Symbol. Executing this command will cause a series of prompts to appear to identify the User Graphic Symbol to be moved, a "base point" for the symbol, and the new location. *Note -* The graphic symbol "location" grip may also be used to move the graphic symbol location.
  Toolbar: User Graphic Commands
  GDI Command Line: MOVESYMBOL, MOVES

Move User Table - Moves a User Table. Executing this command will cause a series of prompts to appear to identify a feature in the User Table to be moved, a "base point" for the User Table, and the new location.
  Toolbar: None
  GDI Command Line: MOVETABLE

Move User Text - Moves a User Text feature. Executing this command will cause a series of prompts to appear to identify the User Text to be moved, a "base point" for the text, and the new location. *Note -* The text "location" grip may also move the test location.
  Toolbar: User Text Commands
  GDI Command Line: MOVETEXT, MOVET

Multiply Customer Loads - Multiplies all the customer *Per Unit Load* values by a specified factor. Executing this command will cause a prompt to appear to enter the Multiplication Factor.
> *Toolbar:* None
> *GDI Command Line:* MULTIPLYCUSTLOAD, MULTICLOAD

Multiply Node Loads - Multiplies all the node *Base Load* values by a specified factor. Executing this command will cause a prompt to appear to enter the Multiplication Factor.
> *Toolbar:* None
> *GDI Command Line:* MULTIPLYNODELOAD, MULTINLOAD
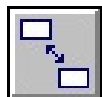
Pan GDI Image - Repositions the GDI Image based on a User-specified location. Executing this command will cause a series of prompts to appear to identify a base point on the GDI Image and the new location. *Note -* The mouse wheel and Auto Pan & Zoom features can also pan the image.
> *Toolbar:* View Controls
> *GDI Command Line:* PANIMAGE, PAN

Plot GDI Image - Plots the current GDI Image. Executing this command will open the Plot Settings screen. Make the required changes, then click the *Plot* command button.
> *Toolbar:* Utility Commands
> *GDI Command Line:* PLOT

Project Coordinates - Projects the model from longitude and latitude into plane coordinates, or vice versa, based on values provided in a specified projection file. Executing this command will open the Convert To Projected Coordinate System screen. Make the required changes, then click the *Continue* command button to change the coordinates.
> *Toolbar:* GeoMap Functions
> *GDI Command Line:* PROJECTCOORDS, PRJ

Purge Customers - Purges (deletes) customer features with invalid coordinate values. Executing this command will cause a message to be displayed asking whether to delete all of the customers with zero or out-of-range coordinates - respond appropriately. If the *Yes* command button is clicked, a message will be displayed stating how many customer records were deleted.
> *Toolbar:* None
> *GDI Command Line:* PURGECUSTOMER, PURGEC

Purge User Text - Purges (deletes) User Text features with invalid coordinate values. Executing this command will cause a message to be displayed asking whether to delete all of the User Text features with zero or out-of-range coordinates - respond appropriately. If the *Yes* command button is clicked, a message will be displayed stating how many User Text records were deleted.

    Toolbar: None
    GDI Command Line: PURGETEXT, PURGET

Quick Size - Performs the solution and pipe sizing routines for the model using the current sizing specification. See the Quick Size Specification screen description for more information.

    Toolbar: None
    GDI Command Line: QUICKSIZE, QSIZE

Quick Solve - Executes the solution routine using the currently set solution options and values. Executing this command will cause the model to be solved using the current Solution Data values, the Solution Data screen will not be displayed, however the solution results will be updated upon completion.

    Toolbar: None - Icon located in the lower-right corner of the GDI.
    GDI Command Line: QUICKSOLVE, QSOLVE

Quick Text - Adds a User Text feature based on the most previously used *Add User Text* values. Executing this command will cause a series of prompts to appear to identify the text location and the text value.

    Toolbar: None
    GDI Command Line: QUICKTEXT, QTEXT

Quick Trace - Performs a Trace based on the most previously used Trace Specifications. Executing this command will cause a prompt to appear to select the starting feature.

    Toolbar: None
    GDI Command Line: QUICKTRACE, QTRACE

Reassign Customer Supply Main - Reassigns a customer feature to a different supply main. Executing this command will cause a series of prompts to appear to identify the customer to reassign and the new supply main for that customer. *Note -* The customer service "tap" grip may also move the test location.

    Toolbar: Customer Commands
    GDI Command Line: REASSIGNCUSTOMER, REASSIGNC

Reassign Customer Supply Main Numbers - Reassigns the supply main (pipe) numbers for all customer features. Executing this command will cause a message to be displayed stating the results of the reassignment. *Note -* Use of this command may corrupt the automatic External Load accounting. Particular care should be taken when using this command.

> *Toolbar:* None
> *GDI Command Line:* REASSIGNSUPPLYNUMBER

 Redo (Restore Last Undo) - Will redo the last "undo." Executing this command will cause the most recent "undo" to be "redone." *Note -* The *Allow Undo Of Data/Graphic Changes* preference settings option must be selected to allow this command to be functional. Also, changes made by some GDI Commands cannot be undone.

> *Toolbar:* Undo/Redo Commands
> *GDI Command Line:* REDO

Reduce Nodes - Eliminates hydraulically unneeded nodes. Executing this command will open the Node Reduction Specifications screen. Make the required changes, then click the *Reduce* command button. The pipe features in the model will be searched for a node between two pipe features of the same size. When this is encountered, and the combined length has not exceeded the specified limit, the node will be deleted, and the pipes will be "joined."

> *Toolbar:* None
> *GDI Command Line:* REDUCENODE, REDUCE

 Regenerate Model Data - Rereads the model data and background images from the disk file. Executing this command will cause the model data and background image data to be reread from the associated disk file. *Note -* This command is useful to restore the original model conditions without closing and reopening the model; however, beware that changes made since the model was last saved will be lost.

> *Toolbar:* GDI Window Controls
> *GDI Command Line:* REGENERATE

Remove Feature From Group - Removes a feature from an existing group. Executing this command will cause a series of prompts to appear to identify the features to be removed from the group.

> *Toolbar:* None
> *GDI Command Line:* REMOVEGROUP, RMVGRP

Remove Fittings - Removes attached fittings from a pipe-type feature. Executing this command will cause a prompt to appear to identify the pipe to remove the fittings.

> *Toolbar:* None
> *GDI Command Line:* REMOVEFITTINGS, RMVFIT

Renumber Nodes - Renumbers the model's node Names based on User-specified criteria. Executing this command will cause the Renumber Specifications screen to be displayed. Enter the desired values, then click the *Renumber* command button.

> *Toolbar:* None
> *GDI Command Line:* RENUMBERNODES, RENUMBER

Repaint GDI Image - Repaint (redraw) the current model image. Executing this command will cause the model and background images to be redrawn in the GDI Display. *Note -* This command is useful for updating the model image after various display modifying commands have been executed when the *Automatically Repaint GDI Image* graphic settings option is not selected.

> *Toolbar:* GDI Window Controls
> *GDI Command Line:* REPAINTGDI, REPAINT

Report Customers With Invalid Coordinates - Reports customer features with large out-of-range or negative coordinate values. Executing this command will open the Customer Coordinate Error Report, listing the customer records with invalid coordinate values.

> *Toolbar:* None
> *GDI Command Line:* REPORTBADCOORD, REPORTBADCUST

Reset Customer Symbol Colors - Resets the color of the customer symbols to their currently saved values. Executing this command will change the color for all of the customer symbols to the currently saved values.

> *Toolbar:* None
> *GDI Command Line:* RESETCUSTOMERCOLOR, RESETCCOLOR

Reset Feature Colors - Resets the colors for all model features (customer symbols, node symbols, pipe lines, and pipe symbols) to their currently saved values. Executing this command will change the color for all model features to their currently saved values.

> *Toolbar:* None
> *GDI Command Line:* RESETCOLORS, RESET

Reset Node Symbol Colors - Resets the color of the node symbols to their currently saved values. Executing this command will change the color for all of the node symbols to the currently saved values.

> *Toolbar:* None
> *GDI Command Line:* RESETNODECOLOR, RESETNCOLOR

Reset Pipe Colors - Resets the color of the pipe lines and symbols to their currently saved values. Executing this command will change the color for all pipe lines and pipe symbols to the currently saved values.
> *Toolbar:* None
> *GDI Command Line:* RESETPIPECOLOR, RESETPCOLOR

Reset Tool Palette - Resets the Tool Palette by removing all of the GDI Icons and returns the Tool Palette to a blank (empty) state. Executing this command will remove all GDI Commands from the Tool Palette.
> *Toolbar:* Utility Commands
> *GDI Command Line:* RESETTOOL
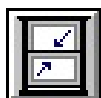
Resize GDI Window - Resizes the Graphic Data Interface (GDI) Window to its default size. Executing this command will cause the size of the GDI to be set to a predetermined height and width.
> *Toolbar:* GDI Window Controls
> *GDI Command Line:* RESIZEGDI, RESIZE

Save Current View - Saves the current view in the GDI Display to the view list. Executing this command will cause a prompt to appear to enter the name for the saved view.
> *Toolbar:* View Controls
> *GDI Command Line:* SAVEVIEW

Save Feature Colors - Saves the colors of its features, which have been temporarily set using either the "Color By" or Trace routines. Executing this command will cause a message to be displayed after the current feature colors have been saved.
> *Toolbar:* None
> *GDI Command Line:* SAVECOLORS

Scale Background Image - Adjusts the Scale Factor required to make the background image match a specified linear distance. Executing this command will cause a series of prompts to appear to identify the background image file to be affected, the known points, and the actual distance. A message will be displayed stating the calculated Scale Factor and asking whether to apply the factor and respond appropriately.
> *Toolbar:* None
> *GDI Command Line:* SCALEBACKGROUND, SCALEBACK

Segment Pipe - Divides a 2-point pipe feature or an arc pipe feature into a User-specified number of individual pipe segments. Executing this command will cause a series of prompts to appear to identify the pipe to segment and the number of pipe segments.

> *Toolbar:* None
> *GDI Command Line:* SEGMENTPIPE

Set Flat Isometric Coordinates - Allows for an isometric style detail to be drawn while in plan view. Executing this command will cause any new lines to be added to the model to be drawn along the three traditional isometric axes.

> *Toolbar:* None
> *GDI Command Line:* SETFLATISO

Set Intersection Flag - Turns the display of the intersection flag for a selected node "on." Executing this command will cause a prompt to appear to identify the associated node. If an intersection flag is not displayed, a flag will now be displayed.

> *Toolbar:* None
> *GDI Command Line:* SETINTERSECTIONFLAG, SETIFLAG

Set Orthogonal Coordinates - Turns the Orthogonal Coordinate entry method "on" or "off" depending on the current status. A message will be displayed stating the current status of the Orthogonal Coordinates. When the Orthogonal Coordinates are turned on, pipe entry is restricted to directions aligned with the X-Y axis.

> *Toolbar:* None
> *GDI Command Line:* SETORTHO, ORTHO

Set Orthogonal Coordinates - Plus - Turns the use of the Orthogonal Plus Coordinate entry method "on" or "off" depending on the current status. A message will be displayed stating the current status of the Orthogonal Plus Coordinates. When the Orthogonal Plus Coordinates are turned on, pipe entry is restricted to directions aligned with the X-Y axis and (the "plus" part) to 45-degree angles to that axis.

> *Toolbar:* None
> *GDI Command Line:* SETORTHOPLUS, ORTHOP

 Set Pipe Attribute Marker - Sets an existing polyline pipe vertex as a pipe attribute marker. Executing this command will cause a series of prompts to appear to identify the pipe associated with the vertex to be changed and the vertex to be set as an attribute marker. This command is only available in the UKDM version of the GASWorkS software.

> *Toolbar:* Graphic Edit Commands
> *GDI Command Line:* SETPIPEATTRIBUTE, SETPATT

Set Service Attribute Marker - Sets an existing service vertex as a service attribute marker. Executing this command will cause a series of prompts to appear to identify the customer associated with the vertex to be changed and the vertex to be set as an attribute marker. This command is only available in the UKDM version of the GASWorkS software.

> *Toolbar:* Customer Commands
> *GDI Command Line:* SETSERVICEATTRIBUTE, SETSERVATT

Set Service Governor Location - Sets an existing service vertex as a governor location marker. Executing this command will cause a series of prompts to appear to identify the customer to be changed, the vertex to be set as a governor location, and additional prompts for information about the governor. This command is only available in the UKDM version of the GASWorkS software.

> *Toolbar:* None
> *GDI Command Line:* SETSERVICELOC

Set User Graphic Line Size (All) - Sets the pattern spacing (display size) for all User Graphic Lines. Executing this command will cause a prompt to appear to select how to change the pattern spacing and the new pattern spacing size value. *Note* - The *Edit Multiple User Graphic Lines* command can also be used to change settings for multiple lines.

> *Toolbar:* None
> *GDI Command Line:* SETLINESIZE

Set User Graphic Symbol Size (All) - Sets the symbol size (display size) for all User Graphic Symbols. Executing this command will cause a prompt to appear to select how to change the symbol size and the new symbol size value. *Note* - The *Edit Multiple User Graphic Symbols* command can also change settings for multiple lines.

> *Toolbar:* None
> *GDI Command Line:* SETSYMBOLSIZE

Set User Text Size (All) - Sets the display size for all User Text features. Executing this command will cause a prompt to appear to select how to change the User Text size and the new User Text size value. *Note* - The *Edit Multiple User Text* command can also be used to change settings for multiple lines.

> *Toolbar:* None
> *GDI Command Line:* SETTEXTSIZE

Set Valve Node - Turns an existing node into a valve node. Executing this command will cause a prompt to appear to select the node to change.

> *Toolbar:* None
> *GDI Command Line:* SETVALVENODE, SETVN

Shift Model Coordinates - Allows the model coordinates to be shifted, scaled, or rotated based on User-specified values. Executing this command will open the Shift Specifications screen. Make the required changes, then click the *Shift* command button.
> *Toolbar:* None
> *GDI Command Line:* SHIFTCOORDS, SHIFT

Show Route Profile - Displays a profile of the various data values along a User-selected route. Executing this command will open the Route Profile Options screen. Make the required changes, then click the *Continue* command button. Depending on the *Selection Style*, one or more additional prompts may be displayed - respond appropriately. The Route Profile screen is displayed. Select the desired tab to view.
> *Toolbar:* None
> *GDI Command Line:* ROUTEPROFILE, PROFILE

Snap Grid Intersections - Turns the Grid Snap "on" or "off" depending on the current status. A message will be displayed stating the current status of the Grid Snap. When the Grid Snap is turned "on" during pipe entry, GASWorkS will snap new nodes and vertices to the closest Reference Grid intersection. The Reference Grid must be displayed (Graphic Settings screen).
> *Toolbar:* None
> *GDI Command Line:* SNAPGRID

Snap Loose Pipe Ends - Snaps pipe ends together where the ends fall within a User-specified *Fuzzy Tolerance* value. Executing this command will open the Fuzzy Tolerance screen. Enter an appropriate value, then click the *Continue* command button. (Pipe ends within this Fuzzy Tolerance will be snapped together.) A message will be displayed stating how many pipe ends were changed.
> *Toolbar:* None
> *GDI Command Line:* SNAPPIPES
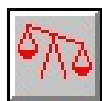
Solve Model - Executes the solution of the model. Executing this command will open the Solution Data screen. Make the required changes, then click the *Solve* command button. The Solution Log will be displayed when the solution is complete.
> *Toolbar:* None - Icon located in the lower-right corner of the GDI.
> *GDI Command Line:* SOLVE

Straighten Polyline Pipe - Straightens a polyline pipe feature by deleting User-selected vertices. Executing this command will cause a prompt to appear to identify the polyline pipe to straighten and the vertices to remove. *Note* - If all vertices are removed, the polyline pipe feature will become a 2-point pipe feature.
> *Toolbar:* None
> *GDI Command Line:* STRAIGHTENPIPE
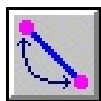
Swap Pipe Ends - Swaps the From Node and To Nodes for a pipe feature. Executing this command will cause a prompt to appear to identify the pipe to swap the ends (nodes) on.
　　　Toolbar: Graphic Edit Commands
　　　GDI Command Line: SWAPPIPEENDS, SWAPENDS

Tap Pipe - Taps (splits or breaks) a pipe feature. Executing this command will cause a series of prompts to appear to identify the pipe to be tapped and the tap location. This command will split a pipe feature into two segments, update the resulting pipe segments, and add a new node at the tap location. Once tapped, a new pipe feature may be extended from the tapped node. *Note -* When entering a new pipe, holding down the *CTRL* key while left-clicking the mouse at a node location near an existing pipe will display a prompt asking whether to tap the existing pipe at that location; respond accordingly.
　　　Toolbar: Graphic Edit Commands
　　　GDI Command Line: PIPETAP, TAP

Text Display Settings - Displays the text display options for the customer, node, and pipe features. Executing this command will open the Text Display Settings screen. Make the required changes, then click the *Apply* command button.
　　　Toolbar: Display Controls
　　　GDI Command Line: SETTEXT
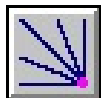
Trace - Performs a Trace and color codes the features along the trace route. Executing this command will open the Trace Specifications screen. Make the required changes, then click the *Trace* command button. Depending on the *Trace Style*, additional prompts will be displayed - respond appropriately.
　　　Toolbar: Utility Commands
　　　GDI Command Line: TRACE

Trace & Update Data - Performs a Trace and updates User-specified data values for features found along the trace route. Executing this command will open the Trace & Update Data Specifications screen. Make the required changes, then click the *Trace & Update* command button. Depending on the *Trace Style*, additional prompts may be displayed - respond appropriately.
　　　Toolbar: None
　　　GDI Command Line: TRACEUPDATE

Turn Background Image Display Off - Turns "off" the background image that is displayed. Executing this command will cause a prompt to appear to identify the background image to be turned "off." The selected background image will be turned "off." *Note -* To turn a background image "on," use the Background Image Settings screen.
> *Toolbar:* None
> *GDI Command Line:* TURNIMGOFF, IMGOFF

Turn Change Log Off - Turns "off" the Change Log. Executing this command will cause the Change Log to be turned "off." Additional items will not be added to the log until it is turned back "on."
> *Toolbar:* None
> *GDI Command Line:* LOGOFF

Turn Change Log On - Turns "on" the Change Log. Executing this command will cause the Change Log to be turned "on" and ask whether the Change Log Comment prompt is displayed after certain *Edit* Commands. When turned "on," certain changes to the model data will be recorded in the log.
> *Toolbar:* None
> *GDI Command Line:* LOGON

Turn DXF Layer Off - Turns "off" a DXF layer associated with an attached DXF background image file. Executing this command will cause a prompt to appear to identify the DXF entity on the layer to be turned "off." The layer associated with the selected entity will be turned "off." *Note -* To turn a DXF layer "on," use the Background Image Settings screen.
> *Toolbar:* None
> *GDI Command Line:* DXFLAYEROFF, DXFOFF

Turn GeoMap On - Turns "on" the GeoMap functions. Executing this command will display a message asking whether to continue with the GeoMap functions. When turned "on," the GeoMap functions allow the use of geographic map services to help in building or maintaining a model.
> *Toolbar:* None - Icon at the end of the GDI Toolbar list.
> *GDI Command Line:* GEOMAP
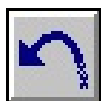
Undo Last Change - Will undo up to the last ten (10) data or graphic changes. Executing this command will cause the most recent data or graphic change to be "undone." *Note -* The *Allow Undo Of Data/Graphic Changes* preference settings option must be selected to allow this command to be functional. Changes made by some GDI Commands cannot be undone.
> *Toolbar:* Undo/Redo Commands
> *GDI Command Line:* UNDO

Unhighlight Group - Unhighlights previously highlighted features in a group. Executing this command will cause the features in a group to be unhighlighted.
>   *Toolbar:* None
>   *GDI Command Line:* UNHIGHLIGHTGROUP, UNHILITGRP

Unset Intersection Flag - Turns the display of the intersection flag for a selected node "off." Executing this command will cause a prompt to appear to identify the associated node. If an intersection flag is already displayed, it will be removed. If an intersection flag is not displayed, a flag will now be displayed.
>   *Toolbar:* None
>   *GDI Command Line:* UNSETINTERSECTIONFLAG, UNSETIFLAG

Unset Pipe Attribute Marker - Removes the attribute marker setting from an existing pipe vertex (attribute marker). Executing this command will cause a series of prompts to appear to identify the pipe associated with the vertex to be changed and the vertex/marker to be unset. This command is only available in the UKDM version of the GASWorkS software.
>   *Toolbar:* Graphic Edit Commands
>   *GDI Command Line:* UNSETPATT, UNSETPIPEATT

Unset Service Attribute Marker - Removes the attribute marker setting from an existing service vertex (attribute marker). Executing this command will cause a prompt to appear to identify the customer associated with the vertex to be changed and the vertex/marker to unset. This command is only available in the UKDM version of the GASWorkS software.
>   *Toolbar:* Customer Commands
>   *GDI Command Line:* UNSETSATT, UNSETSERVICEATT

Unset Valve Node - Turns an existing valve node into a "normal" system node. Executing this command will cause a prompt to appear to select the desired valve node to be unset.
>   *Toolbar:* None
>   *GDI Command Line:* UNSETVALVENODE, UNSETVN

Update Pipe Customer Count - Updates the *Customer Count* value for all pipe features. Executing this command will cause the customer count for all of the pipes to be updated based on proximity to the customer tap locations.
>   *Toolbar:* None
>   *GDI Command Line:* UPDATEPIPECOUNT, UPDPIPE

View Change Log - Displays the Change Log. Executing this command will open the Change Log for viewing or printing. Click the *Close* command button to close the screen before making a change, which might try to be added to the log.
> *Toolbar:* None
> *GDI Command Line:* VIEWLOG

View CSV Table - Displays a previously created CSV file (table). Executing this command will open the File Selection screen. Select a previously saved CSV file to view, then click the *Continue* command button. Review the contents, then click the *Close* command button.
> *Toolbar:* None
> *GDI Command Line:* VIEWTABLE

View Trace Results - Displays a previously created Trace Results file. Executing this command will open the File Selection screen. Select a previously saved Trace Results file to view, then click the *Continue* command button. The Trace Results Report will be displayed.
> *Toolbar:* Utility Commands
> *GDI Command Line:* VIEWTRACE
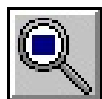
Zoom To Fit - Displays the model image so that the entire model and background image fits within the GDI Display. Executing this command will cause the display to be "zoomed" to fit all of the models and background image in the GDI Display.
> *Toolbar:* View Controls - Icon located in the lower-left corner of the GDI.
> *GDI Command Line:* ZOOMFIT, ZF

Zoom In - Displays the model image so that the view is enlarged in size. Executing this command will cause the display to be "zoomed" in (enlarged) in the GDI Display. *Note* - The amount that the model image changes is determined by the value of the *Zoom Ratio* in the Graphic Settings. *Note* - The mouse wheel may also be used to enlarge the image display.
> *Toolbar:* View Controls - Icon located in the lower-left corner of the GDI.
> *GDI Command Line:* ZOOMIN, ZI

Zoom Model - Displays the model image so that the model (only) image fits within the GDI. Executing this command will cause the display to be "zoomed" to fit all of the model image (customers, nodes, and pipes) in the GDI Display.
> *Toolbar:* None
> *GDI Command Line:* ZOOMMODEL, ZM

Zoom Next - Displays the next view if the *Zoom Previous* command has been used. Executing this command will cause the model image to display the last "zoomed view" before the last use of the *Zoom Previous* command. Up to 10 "zoom views" may be maintained in the "zoom view" list per GDI Panel.
    Toolbar: View Controls
    GDI Command Line: ZOOMNEXT, ZN

Zoom Out - Displays the model image so that the view is reduced in size. Executing this command will cause the display to be "zoomed" out (reduced) in the GDI Display. *Note -* The amount that the model image changes is determined by the value of the *Zoom Ratio* in the Graphic Settings. *Note -* The mouse wheel may also be used to reduce the image display.
    Toolbar: View Controls - Icon located in the lower-left corner of the GDI.
    GDI Command Line: ZOOMOUT, ZO

Zoom Previous - Displays the previous zoomed view in the "zoom view" list. Executing this command will cause the model image to display the previous "zoomed view" in the "zoom view" list. Up to 10 "zoom views" may be maintained in the "zoom view" list per GDI Panel.
    Toolbar: View Controls
    GDI Command Line: ZOOMPREVIOUS, ZP

Zoom To Saved View - Displays a previously saved view. Executing this command causes a prompt to appear to identify the *View Name* to display.
    Toolbar: View Controls
    GDI Command Line: ZOOMVIEW, ZV

Zoom By Scale Factor - Displays the model image "zoomed" by a Scale Factor value. Executing this command will cause a prompt to appear to enter the Scale Factor (zoom scale) value. *Note -* Scale Factors less than one will reduce the model image size. Scale Factors greater than one will enlarge the model image size.
    Toolbar: View Controls
    GDI Command Line: ZOOMSCALE, ZS

Zoom Window - Displays the model image to fit into a User-selected window. Executing this command will cause a series of prompts to appear to identify the corners of the zoom window.
    Toolbar: View Controls
    GDI Command Line: ZOOMWINDOW, ZW

Zoom World Extents - Displays the image to the visible portion of the "world" map from the assigned geographic mapping service. Executing this command will cause the visible portion of the "world" map from the assigned geographic mapping service to be displayed. *Note -* Generally, the map extents are from -180 degrees to 180 degrees longitude and about -85 degrees to 85 degrees longitude.

> *Toolbar:* GeoMap Functions
> *GDI Command Line:* ZOOMWORLD

Mouse Events

While working in the GDI, actions may be triggered by right- or left-clicking the mouse (pressing the right or left mouse button) and clicking or rolling the mouse wheel.

Left-clicking the mouse will cause the nearest customer, node, pipe, or other graphic feature to the cursor to be selected. The data associated with the feature will be displayed in the Data Panel.

Right-clicking the mouse while a GDI Command is executed can cause various actions. If the GDI Command requires a value entry, right-clicking the mouse is the same as pressing the *Enter* key. If the GDI Command requires a feature selection, right-clicking the mouse is the same as left-clicking the mouse.

When the *Automatically Pan & Zoom* graphic settings option is selected, certain mouse events will cause the GDI Image to be zoomed or panned. The actions associated with these events are described below:

> ● To pan (move) the model image, hold down the left mouse button or mouse wheel, move the cursor to the new location, and release the mouse button. The model image will move as appropriate.

> ● To zoom in by a window (make a specified portion of the GDI Image larger), hold down the right mouse button and move the cursor to the upper-left corner of the desired zoom window. When the mouse button is released, the GDI Image will be redrawn as appropriate.

> ● To zoom out (make the GDI Image smaller), hold down the right mouse button and move the cursor to the left and down. When the mouse button is released, the GDI Image will be redrawn proportional to the distance that the cursor was moved compared to the diagonal display distance of the GDI Display.

> ● To zoom in or out using the mouse wheel, place the mouse cursor at the center of the area to be enlarged or reduced, roll the mouse wheel away to enlarge the image, and roll the mouse wheel towards you to reduce the image.

Key Events

While working in the GDI, certain keystrokes will cause various actions. The actions are described in the following table:

Key Event	Resulting Action
Ctrl key + *Z* key	Causes the *Undo* command to be executed.
Ctrl key + *F* key	Causes the *Find Any Feature* command to be executed. At the *Find What* prompt, select the desired feature from the GDI Prompt List. Additional prompts will be displayed and vary by the selected type - respond appropriately.
Delete key	When a data feature is highlighted, pressing the *Delete* key will cause the delete command associated with that feature to be executed.
Enter key	If no GDI Command is being executed, repeats the last GDI Command. Or if a GDI Command prompt is being displayed, causes the currently displayed prompt value to be accepted.
Esc key	Causes the current GDI Command to be canceled (terminated).

Mouse & Key Events

The table below describes mouse and key combinations that will cause the following actions to occur:

Mouse & Key Event	Resulting Action
C key + Left-Click *Ctrl* key + Left-Click	Causes the customer feature nearest the cursor to be selected. The data associated with the selected customer will be displayed in the Data Panel. Similar to executing the *Edit Customer Data* command.
N Key + Left-Click *Shift* Key + Left-Click	Causes the node feature nearest the cursor to be selected. The data associated with the selected node will be displayed in the Data Panel. Similar to executing the *Edit Node Data* command.
P Key + Left-Click	Causes the pipe feature nearest the cursor to be selected. The data associated with the selected pipe will be displayed in the Data Panel. Similar to executing the *Edit Pipe Data* command.
T key + Left-Click *Alt* key + Left-Click	Causes the User Text feature nearest the cursor to be selected. The data associated with the selected User Text will be displayed in the Data Panel. Similar to executing the *Edit User Text* command.

Mouse & Key Event	Resulting Action
V key + Left-Click	Causes the pipe vertex nearest to the cursor to be selected. The data associated with the selected vertex will be displayed in the Data Panel.
Ctrl key+ Left-Click	When performed during a node location prompt, a prompt will be displayed asking whether to "tap" the pipe nearest to the selected location. If "Yes," the nearby pipe will be tapped, the new node will be placed at the tap location, and the pipe associated with the new node will be snapped to the tap location.

Entering Coordinates

Many of the GDI Commands require a location to be selected. When a coordinate pair is required, the coordinate value may be entered using several methods. The methods are described in the following table:

Entry Method	Entry Instructions
Coordinate Entry	Coordinate values may be entered by typing the desired values for the 2D or 3D values. Separate the individual values with a comma or space character. Do <u>not</u> include a digit group separator when entering the coordinate values. *For Example:* 1000,1000 or 1000 1000 - If using a comma "," as a decimal separator, use a space.
Default	When re-executing one of the *Add Pipe* commands, the coordinate values for the To Node will be displayed as the location for the new From Node. To accept the value, press the *Enter* key or right-click the mouse.
Distance & Relative Angle	Sometimes, a location may be entered using a distance and a relative angle. To use this method, enter the "@" symbol followed by the distance value, then the "<" symbol and the angle value. *For Example:* @500<90 *Note* - The top of the screen represents an angle position of (0) zero. Positive angles are measured clockwise from 12 o'clock (North). *Note* - The distance and relative angle method may be used with the various "*Add Pipe*" commands or the *Add User Graphic Line* command to specify the Next End, To Node, or vertex location.

Entry Method	Entry Instructions
Fixed Length & Selected Angle	Sometimes, a location may be entered using a fixed length and a graphically selected angle. To use this method, enter the "@" symbol followed by the distance value and then "<?." *For Example:* @500<?. At the *Pick A Point To Establish The Angle* prompt, select the desired angle. *Note* - The fixed length and selected angle method may be used with the various "*Add Pipe*" commands or the *Add User Graphic Line* command to specify the Next End, To Node, or vertex location.
Mouse Selection	Freestyle locations may be entered by moving the mouse crosshairs to the desired location and left-clicking the mouse to use the selected location.
Node Name	The location of an existing node may be used by entering the node's Name (number) at the GDI Prompt. To use this method, enter a "N" followed by the name of the node to be used. *For Example:* N 25 or N25.
Snap To An Existing Node (Using Auto Snap)	A location may be snapped to an existing node location using the Auto Snap feature. The Auto Snap feature causes the coordinates of the nearest node to the mouse crosshairs to be used. To use the Auto Snap feature, hold down the *Shift* key while left-clicking the mouse.
Snap To An Existing Node (Using Feature Snap)	A location may be snapped to an existing node using the Feature Snap. The Feature Snap causes the coordinates of a node falling within the Snap Target of the mouse crosshairs to be used. To turn "on" the Feature Snap, click on the *Feature Snap* icon.

When entering coordinates for a new node's location, if an existing node is found within a distance equal to the Snap Target, the new node's coordinates are ignored, and the existing node's coordinates are used. This prevents multiple nodes from being placed at or near the same location. Additionally, the *Feature Snap* command described earlier in this section may be used to force node snapping when entering new coordinate values.

When using two-dimensional (2D) coordinates, the "X" and "Y" coordinates represent locations in the horizontal plane and are usually viewed from above using the "plan" view. Elevation values may be entered for node and vertex locations. However, no "Z" coordinate value is used.

When using three-dimensional (3D) coordinates, the "X" and "Y" coordinates represent locations in the horizontal plane, and the "Z" coordinate represents a location above or below the horizontal plane. Models using 3D coordinates are usually viewed using the "isometric" view. Elevation values may be entered for node and vertex locations and may differ from the "Z" coordinate.

When using "world" coordinates (one of the degree longitude and latitude dimensional units - "Degree L/L" or "Degree Web L/L"), coordinate values must be entered as longitude ("X") and then latitude ("Y"). Values must be entered in decimal degrees with six digits to the right of the decimal place.

Change Log

The Change Log is accessed by executing the *View Change Log* command. The features associated with the log are described as follows.
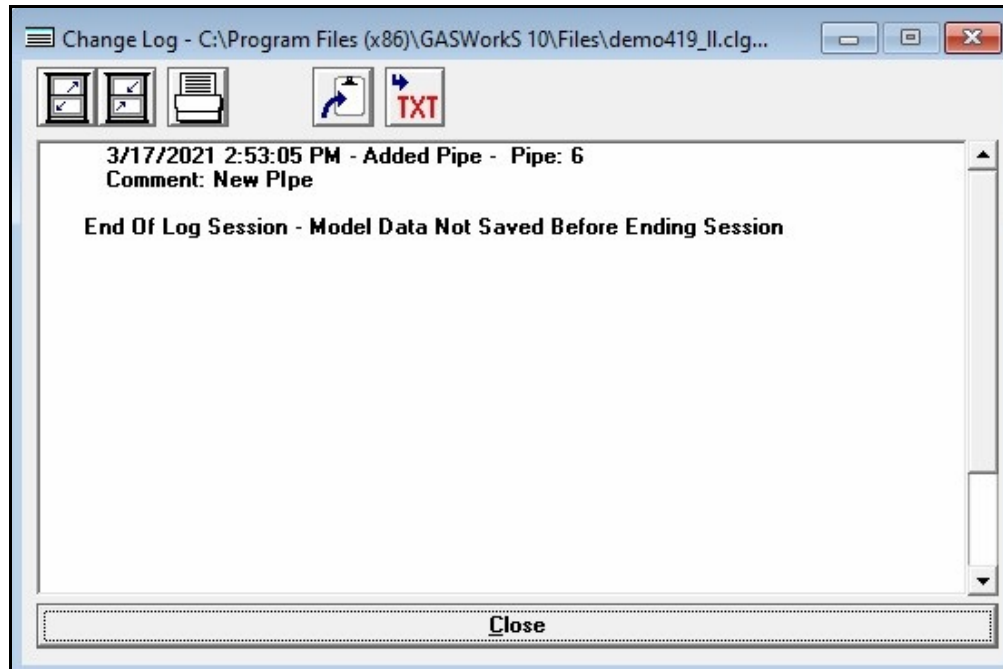
Displayed Items

Change Log - Displays the Change Log. To browse the display, adjust the vertical scroll bar adjacent to the display area.

Icons

 Copy To Clipboard - Copies the contents of the Change Log to the Windows Clipboard. Once copied, the contents may be pasted into another application.
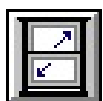
 Maximize Screen Size - Increase the Change Log screen size to fill the GASWorkS display.

Print - Prints the contents of the Change Log. When clicked, the Print Settings screen is displayed.
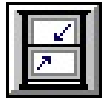
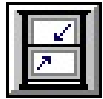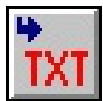**Reset Screen Size -** Sets the Change Log screen's size to its "normal" dimensions and position.
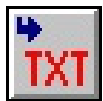
Save To Text File - Saves the data of the Change Log contents to a text file (.txt). When clicked, the File Selection screen is displayed.

Command Buttons

Close - Closes the Change Log.

Notes & Considerations

● The Change Log is a useful way to keep track of revisions and changes made to a model, particularly during the building phase. It tracks certain changes to the model, such as added or deleted pipes, nodes, or customers. The record for each change includes a description and allows a comment to be added if desired.

● The Change Log tracking may be turned on or off using the *Turn Change Log On* or *Turn Change Log Off* GDI commands.

Color Display Settings Screen

The Color Display Settings screen is accessed by executing the *Color Display Settings* command or selecting the *Color Display Settings* item from the *Settings & Actions* menu of the *Data Panel*. The screen allows the display color of the basic GDI features to be set by the User. Using this screen, colors may be set by three methods - by overall default colors, by query selection of a specific data value, or by range assignment. The features associated with the screen are described as follows.

Default Colors Tab
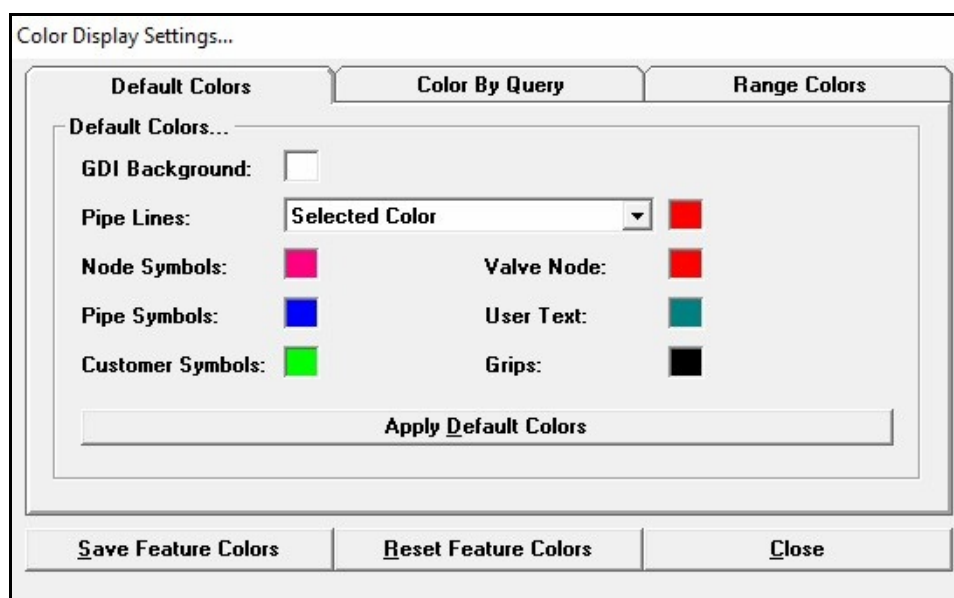
Data Items

Customer Symbols - Specifies the "default" color to display the customer symbol features.

GDI Background - Specifies the color of the background of the GDI Display.

Grips - Specifies the "default" color to display the grips.

Node Symbols - Specifies the "default" color to display the node symbols.

Pipe Lines - Specifies the "default" color to display the pipe line features.

Pipe Symbols - Specifies the "default" color to display the pipe symbols.

GASWorkS™ 10.1

GDI

User Text - Specifies the "default" color to display the User Text features.

Valve Node - Specifies the "default" color to display the valve node features.

Command Buttons

Apply Default Colors - Applies the current default color settings to the displayed image.

Color By Query Tab

This section may apply a highlight color to a selected set of customer, node, or pipe features based on a User-specified selection criteria.

```
Color Display Settings...
┌──────────────────────┬──────────────────────┬──────────────────────┐
│   Default Colors     │    Color By Query    │     Range Colors     │
├──────────────────────┴──────────────────────┴──────────────────────┤
│ ┌─ Color By Query... ──────────────────────────────────────────┐   │
│ │  Color: [yellow]  ┌──────────────────────────────────────┐   │   │
│ │                   │      Define Query Specifications     │   │   │
│ │                   └──────────────────────────────────────┘   │   │
│ │                                                              │   │
│ │                                                              │   │
│ │                                                              │   │
│ │                   ┌──────────────────────────────────────┐   │   │
│ │                   │         Apply Query Colors           │   │   │
│ │                   └──────────────────────────────────────┘   │   │
│ └──────────────────────────────────────────────────────────────┘   │
│ ┌──────────────────┬──────────────────────┬──────────────────┐     │
│ │ Save Feature Colors │ Reset Feature Colors │    Close       │     │
│ └──────────────────┴──────────────────────┴──────────────────┘     │
└──────────────────────────────────────────────────────────────────────┘
```
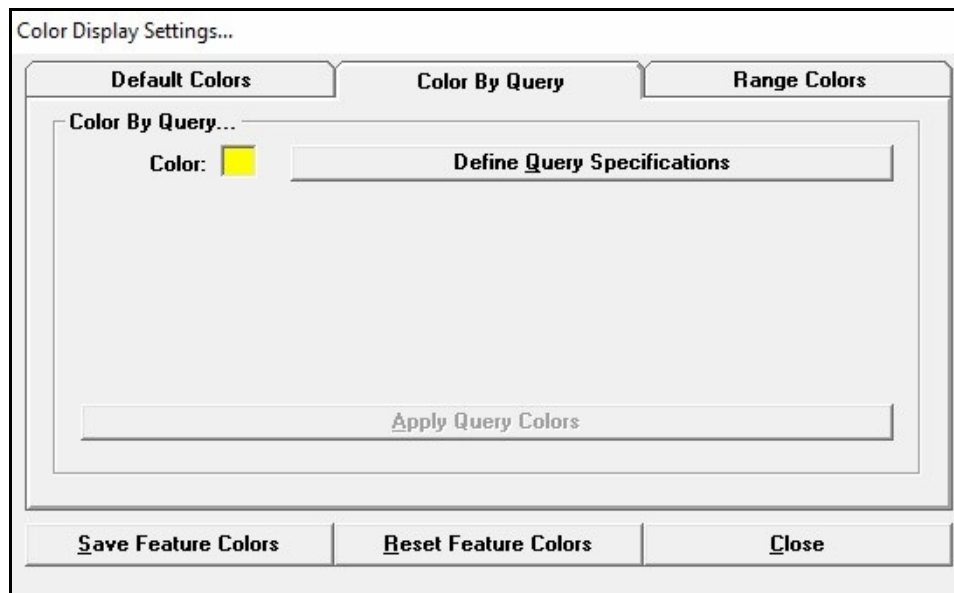
Data Items

Color - Specifies the color used to display the features found by the Query Specifications.

Command Buttons

Apply Query Colors - Applies the selected color to the features in the Query Specifications.

Define Query Specifications - Displays the Query Specifications screen to define the query specifications. A selection set may be defined by specifying which features to assign the specified color. See the *Query Specifications* topic in the *Data Screens* section for a complete screen description.

Range Colors Tab

This section may color code customer, node, and pipe features based on User-specified range criteria. The section consists of a group of color panels and range fields. Each color panel is associated with the adjacent value range. The range values may be manually set by the User or automatically set using either the *Allocate* or *Calculate* command buttons.

Displayed Items

From - Displays the "From" value of the associated range. The "To" value from the preceding range is automatically set as the "From" value for the following range. The "From" values cannot be directly changed by the User.

Data Items

Color Panels - Specifies which colors will display the features associated with the adjacent range.

Item - Specifies which data item to associate with the specified ranges.

To - Specifies the "To" value of the associated range. The "To" values may be set by two methods. One is to automatically set the values using the "allocate" and "calculate" methods. The other is to manually enter the value.

Units - Specifies the dimensional units to use during the "allocate" or "calculate" methods. The *Units* list is only displayed when an appropriate item is selected from the *Item* list.

Command Buttons

Allocate - Automatically establishes the range values using the "allocate" method. The "allocate" method attempts to create ranges with approximately the same number of features in each range group. The "allocate" method does not always use all range colors.

Apply Range Colors - Applies the color range values to the associated features.

Calculate - Automatically establishes the range values using the "calculate" method. The "calculate" method creates eight evenly "spaced" ranges based on the overall value range of the selected item. Not all colors may be displayed with this method; if there are no features that fall within a range, no color will be displayed.

Command Buttons - *General*

Close - Closes the screen.

Reset Feature Colors - Resets the feature colors to the values currently saved in the model data files.

Save Feature Colors - Saves the currently displayed colors.

Notes & Considerations

● To set a Color, click in the associated box. The Color Palette will be displayed. Select the desired color, then click the *OK* command button to assign the selected color.

● Features are considered to be within a range if the value associated with the feature is greater than the "From" value and less than or equal to the "To" value for the range.

● The pipe default colors may be set by several methods, including:
  • To a specific color.
  • To values assigned in the assigned Pipe Property Table.
  • To values assigned in the assigned Facility Table.

● If the color for a data feature is set to a specific value, the default color settings will not affect its display.

● The values applied using the Color Display Settings screen only affect the currently displayed GDI Display panel.

GASWorkS™ 10.1 *GDI*

Color Legend

When a data item is color-coded using the Color By Range routine, a legend of the colors and associated ranges may be displayed using the Color Legend. The features associated with the screen are described as follows.

🟥	Below -3835.072
🟨	-3835.072 <= -3287.072
🟩	-3287.072 <= -2739.072
	-2739.072 <= -2191.072
🟦	-2191.072 <= -1643.072
	-1643.072 <= -1095.072
🟩	-1095.072 <= -547.072
⬛	Above -547.072

Customer Load Per Unit, ... **Apply**
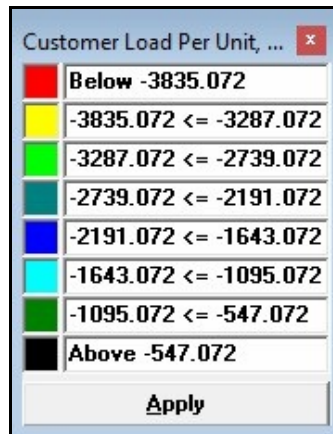
Displayed Items

Color Ranges - Represent the value ranges associated with each display color. These values are automatically assigned and cannot be changed by the User.

Command Buttons

Apply - Closes the screen and reapplies the range colors.

Notes & Considerations

● The location of the Color Legend may be adjusted by dragging and dropping the screen.

● The size of the Color Legend may be adjusted by dragging the edges or corners.

● The Color Legend may be converted to User Text so that it may be plotted. To do this, use the *Create Color Legend Text* command.

Convert To Projected Coordinate System Screen

The *Project Coordinates* command allows a model to be projected from longitude-latitude coordinates into plane coordinates and from plane coordinates to longitude-latitude coordinates. For example, a model using a Geographic Coordinate System (longitude and latitude) may be converted into a model using a State Plane Coordinate System if the appropriate projection file is provided. Executing this command will display the Convert To Projected Coordinate System screen. The features associated with the screen are described as follows.

Displayed Items

Parameters - Displays the parameters of the projected coordinate system from the specified projection file. The panel's contents will vary depending on the projection file selected and the projection type.

Data Items

Conversion - Specifies whether the model is projected from longitude-latitude to plane coordinates or vice versa.

File Type - Specifies the projection file type. If the *Browse* command button is used to select a Filename, only those files with the selected File Type will be displayed.

Filename - Specifies the name of the projection file to be used. If the Filename is manually entered, include the path and extension. Extension values will vary according to the file type.

Update Pipe Lengths - When selected, the length values of all of the pipe features will be recalculated based on the new coordinate values of the nodes and vertices. Do <u>not</u> select this option if the original length values were entered manually or the original values are to be maintained.

Command Buttons

Browse - Used to find and select a previously saved projection file name. When executed, it displays the File Selection screen.

Cancel - Closes the screen without proceeding with the Projection routine.

Continue - Proceeds to project the model coordinates as specified.

Notes & Considerations

● The new coordinate values are calculated in the following order: nodes, vertices, pipes, customers, and other data features.

● To convert from one coordinate projection to another – for example, a Transverse Mercator projection to a Lambert Conformal Conic projection – the model must first be projected into longitude-latitude, then re-projected using the new projection. The Projection routine will attempt to determine whether the model is in longitude-latitude or projected coordinates. If the User attempts a projection using an already projected model or returns to longitude-latitude when the model is already in longitude-latitude coordinates, a message will be displayed.

● The Projection routine is designed to work with ESRI Projection files (.prj) and EPSG Well Known Text files (.wkt). Other file formats are not supported at this time.

● This routine does not project background files. See the *Background Image Settings* topic in the *Data Screens* section for more information on how to project background files.

● Feature sizes like symbol size or text height are adjusted to reflect the new coordinate system.

Customer Coordinate Error Report

The Customer Coordinate Error Report is created and displayed by executing the *Report Customers With Invalid Coordinates* command. The features associated with the report are described as follows.
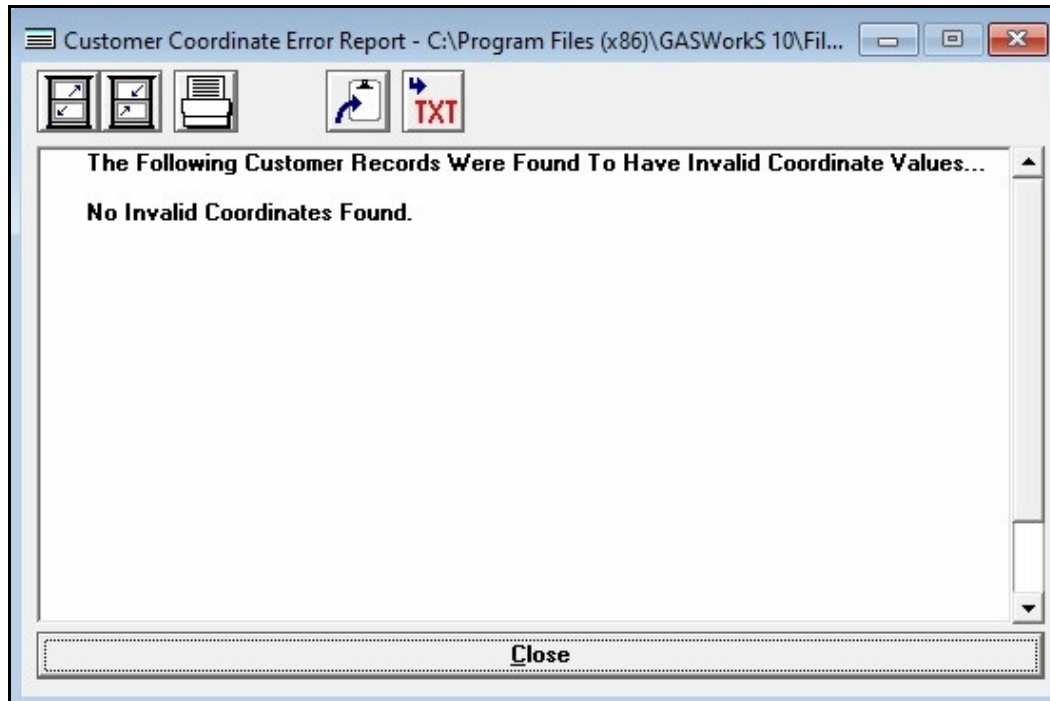
Displayed Items

Customer Coordinate Error Report - Displays the contents of the Customer Coordinate Error Report. To browse the report, adjust the vertical scroll bar adjacent to the display area.

Icons

 Copy To Clipboard - Copies the contents of the Customer Coordinate Error Report to the Windows Clipboard. Once copied, the contents may be pasted into another application.
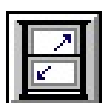
 Maximize Screen Size - Increases the Customer Coordinate Error Report's size to fill the GASWorkS display.

Print - Prints the contents of the Customer Coordinate Error Report. When clicked, the Print Settings screen is displayed.

Reset Screen Size - Sets the Customer Coordinate Error Report's size to its "normal" dimensions and position.

Save To Text File - Saves the data of the Customer Coordinate Error Report contents to a text file (.txt). When clicked, the File Selection screen is displayed.

Command Buttons

Close - Closes the Customer Coordinate Error Report.

Customer Link ID Number Report

The Customer Link ID Number Report is created and displayed when the *Flag Customers - Invalid Link ID Numbers* command or *Flag Customers - Redundant Link ID Numbers* commands are executed. The features associated with the report are described as follows.
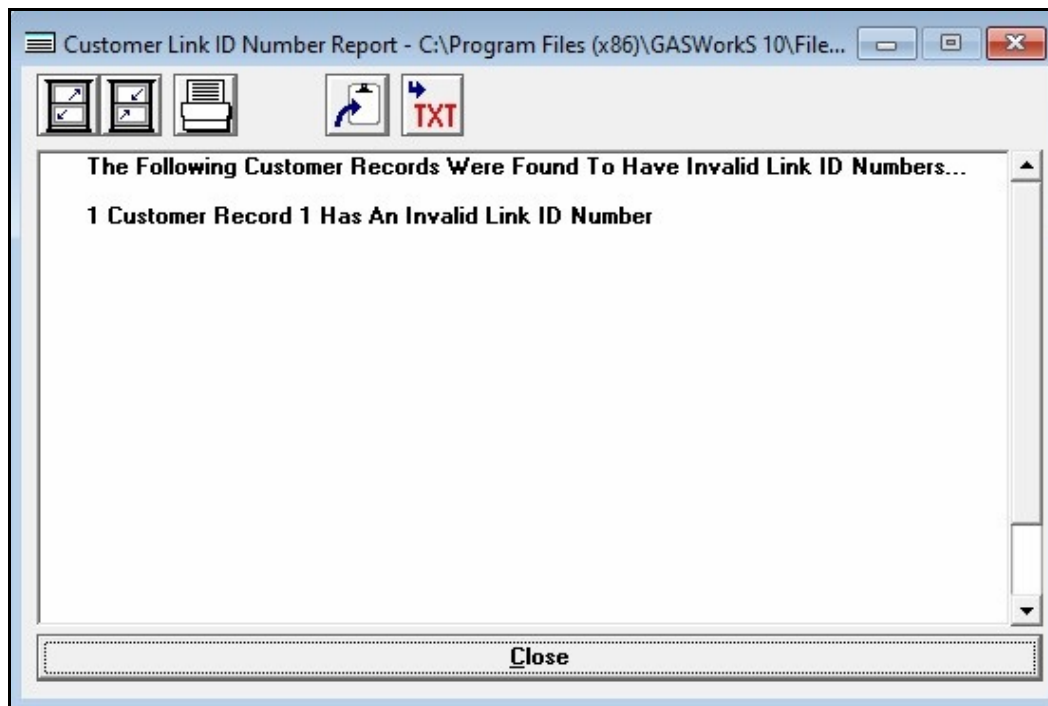
Displayed Items

Customer Link ID Number Report - Displays the contents of the Customer Link ID Number Report. To browse the report, adjust the vertical scroll bar adjacent to the display area.

Icons
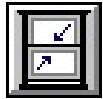
Copy To Clipboard - Copies the contents of the Customer Link ID Number Report to the Windows Clipboard. Once copied, the contents may be pasted into another application.
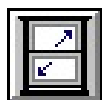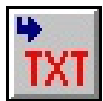
Maximize Screen Size - Increases the Customer Link ID Number Report's size to fill the GASWorkS display.

Print - Prints the contents of the Customer Link ID Number Report. When clicked, the Print Settings screen is displayed.

Reset Screen Size - Sets the Customer Link ID Number Report's size to its "normal" dimensions and position.

Save To Text File - Saves the data of the Customer Link ID Number Report contents to a text file (.txt). When clicked, the File Selection screen is displayed.

Command Buttons

Close - Closes the Customer Link ID Number Report.

Notes & Considerations

● The *Flag Customers - Invalid Link ID Number* command can be used to identify customers where the assigned Link ID value does not appear in the attached Customer Attribute file/database. Discovered customers are listed in the Customer Link ID Number Report.

Edit User Text Value Screen

The Edit User Text Value screen is used to edit a User Text value. The screen is displayed when the *Text Value* cell is clicked when a User Text feature is displayed in the Data Panel. The features associated with the screen are described as follows.
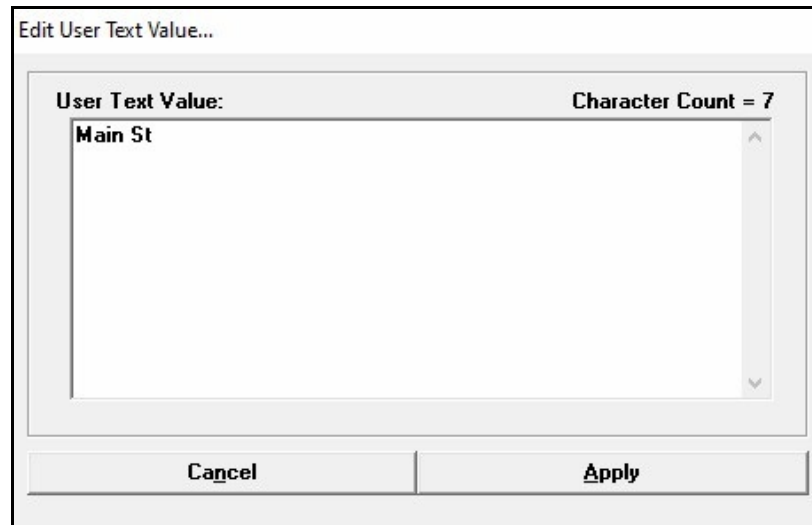
Displayed Items

Character Count - Displays the number of characters in the User Text value. *Note -* Up to 4096 characters are allowed.

Data Items

User Text Value - Specifies the User Text value to display. To edit the value, use basic text edit functions by typing in the panel.

Command Buttons

Apply - Closes the screen and applies changes.

Cancel - Closes the screen without saving or applying changes.

Notes & Considerations

● Multi-line text may be created in two methods. The first is by setting the width of the text display to a value greater than zero (0). With this method, text is automatically wrapped when its length is greater than the specified text display width. The second method is to insert a "line break" by pressing the *Enter* key at locations where text lines are to end. The two methods may be combined.

● When the Text Display Style is set to Single Line, and line breaks are included in the text, symbols will be displayed at the location of the line breaks.

Fuzzy Tolerance Screen

Several commands require a *Fuzzy Tolerance* value to be set. When these commands are executed, the Fuzzy Tolerance screen is displayed. The features associated with the screen are described as follows.
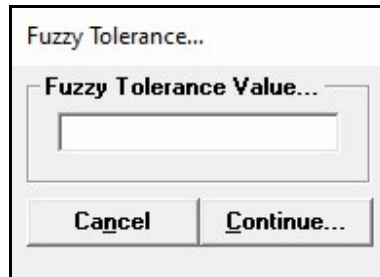
Data Items

Fuzzy Tolerance Value - Specifies the *Fuzzy Tolerance* value.

Command Buttons

Continue - Proceeds with the associated routine.

Notes & Considerations

● The Fuzzy Tolerance screen is displayed by several commands. The value is generally used to search for a feature within the specified value of another feature's location. For example, to search for a pipe feature within the specified distance of a node feature.

Model Comparison Options Screen

The Model Comparison Report includes several specialized report options. The User may indicate which sections to include in the Model Comparison Report. The Model Comparison Options screen is accessed by selecting the *Compare Two Models* item from the GDI Command List. The features associated with the screen are described as follows.
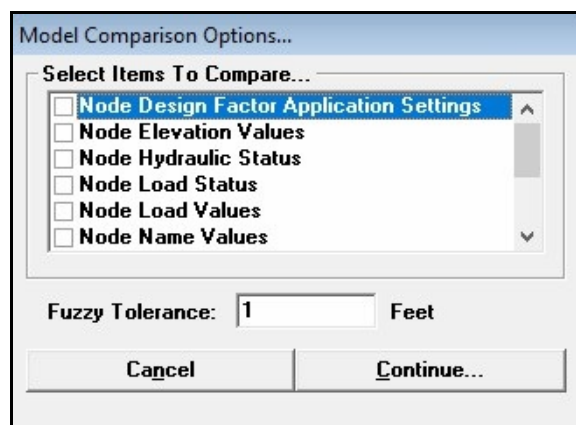
Data Items

Fuzzy Tolerance - Specifies the *Fuzzy Tolerance* value when comparing the two models. The value is expressed in the *Coordinates* units.

Node (Data Items) **-** When selected, the value associated with the selected node data item will be compared between the currently open model and the selected model. If a value difference is found, a message is included in the Model Comparison Report.

Node XY Locations - When selected, the X-Y location of the nodes in each model is checked to see if a node exists at that same location (within the specified Fuzzy Tolerance) in the other model. If a coincident node is not found, a record is included in the Model Comparison Report.

Pipe (Data Items) **-** When selected, the value associated with the selected pipe data item will be compared between the currently open model and the selected model. If a value difference is found, a message is included in the Model Comparison Report.

Pipe XY Locations - When selected, the X-Y location of the end nodes and vertices for each pipe in each model is checked to see if an identical pipe exists at the same locations (within the specified Fuzzy Tolerance) in the other model. If a coincident pipe is not found, a message is included in the Model Comparison Report.

Command Buttons

Cancel - Closes the screen without proceeding with the Model Comparison routine.

Continue - Proceeds with the Model Comparison routine.

Notes & Considerations

● The model comparison routine checks the selected values and locations between models. If a difference is found, it is added to the reported issues. The routine is useful when it is unclear what changes have been made between an older model and a newer one.

GASWorkS™ 10.1 — *GDI*

Model Comparison Report

The Model Comparison Report includes several specialized report sections selected by the User. These sections may include a comparison of various node and pipe values. The User may indicate which sections to include in the report using the Model Comparison Options screen. The report is accessed by selecting the *Compare Two Models* item from the GDI Command List. The features associated with the report are described as follows.
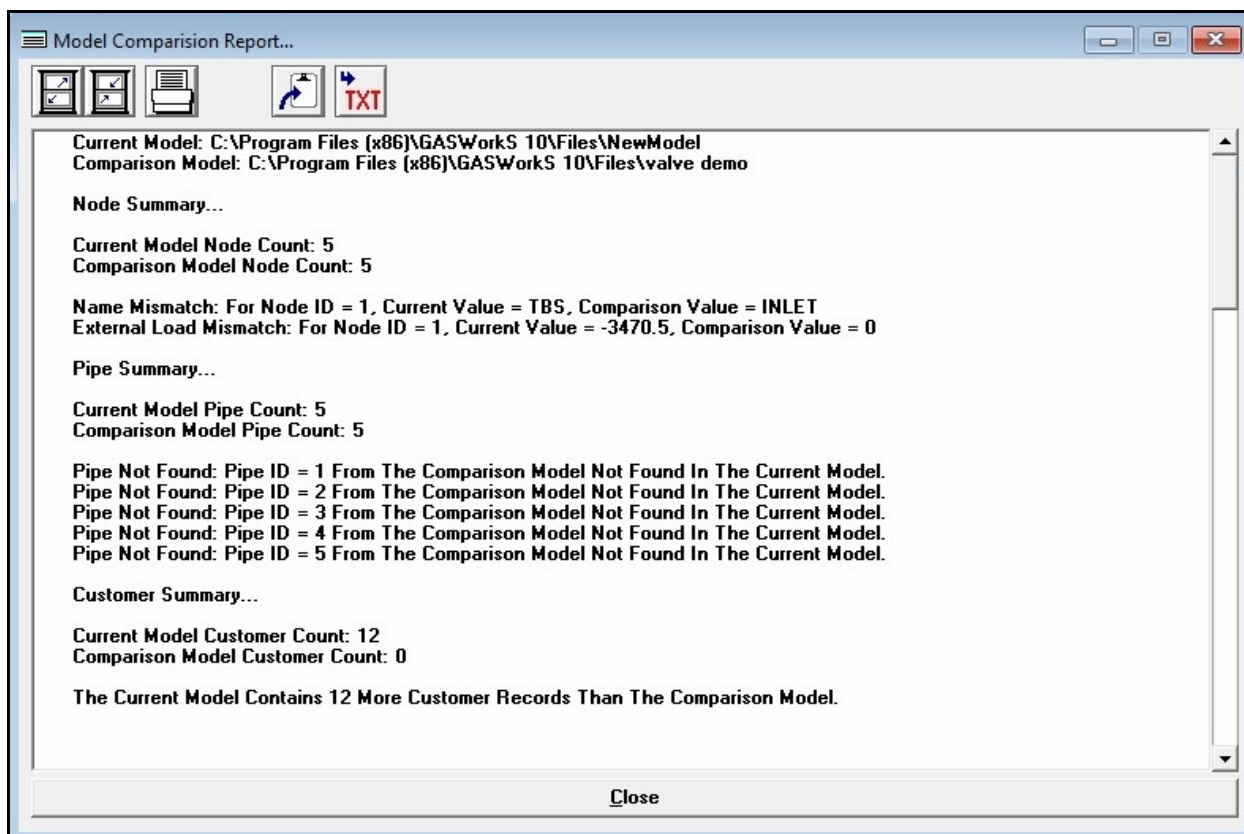
Displayed Items

Model Comparison Report - Displays the contents of the Model Comparison Report. To browse the report, adjust the vertical scroll bar adjacent to the display area.

Icons

Copy To Clipboard - Copies the contents of the Model Comparison Report to the Windows Clipboard. Once copied, the contents may be pasted into another application.
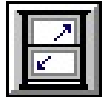
Maximize Screen Size - Increases the Model Comparison Report's size to fill the GASWorkS display.

Print - Prints the contents of the Model Comparison Report. When clicked, the Print Settings screen is displayed.
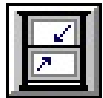
Reset Screen Size - Sets the Model Comparison Report's size to its "normal" dimensions and position.
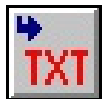
Save To Text File - Saves the data of the Model Comparison Report contents to a text file (.txt). When clicked, the File Selection screen is displayed.

Command Buttons

Close - Closes the Model Comparison Report.

Notes & Considerations

● The report lists differences found by the Model Comparison routine.

● See the *Model Comparison Options Screen* topic in the *Graphic Data Interface (GDI)* section for more information on selecting the items that may be reported.

Multiple Customers Found Screen

The Multiple Customers Found screen is displayed when there is more than one customer feature found at or near the same location (customer symbol) when selecting a customer feature for editing. The features associated with the screen are described as follows.
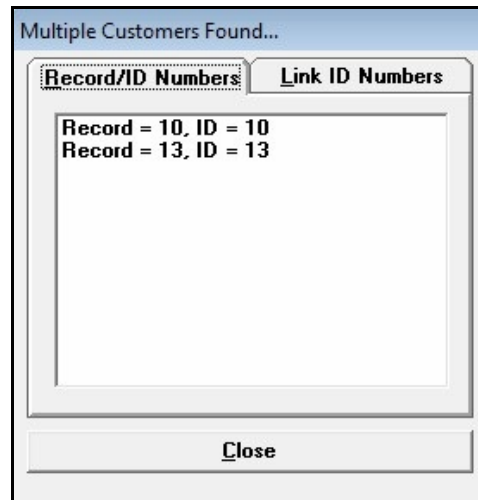
Displayed Items

Record/ID Numbers - Displays the Record Numbers associated with the customer features found at the selected location. Double-click on a number to view or edit the associated customer's data.

Link ID Numbers - Displays the Link ID Numbers associated with the customer features found at the selected location. Double-click on a number to view or edit the associated customer's data.

Command Buttons

Close - Closes the screen.

Notes & Considerations

● Depending on the "zoom" scale, when the command is executed that requires the selection of a customer location, and multiple customers appear to exist at the same location, the Multiple Customers Found screen may appear. When the screen appears, select the desired customer from the "Record/ID Numbers" or "Link ID Numbers" list. When selected, the data for the associated customer will be displayed in the Data Panel. Select the *Close* command button to cancel the selection.

Multiple Edit Specifications Screen

Certain model features may be edited using one of the "*Edit Multiple*" commands. When one of these commands is executed, the desired changes are made using the Multiple Edit Specifications screen. The features associated with the screen are described as follows.

Model Data Items Tab

Data Items

Equal To - Specifies the new value for the selected *Set* item.

Set - Specifies the data item to be changed.

Engineering & Software Services

Commands Tab

Data Items

Command - Specifies the GDI Command to execute.

Attribute Data Items Tab

Multiple Edit Specifications...		
Model Data Items	**Commands**	**Attribute Data Items**

Set...
`Alias` ▾

Equal To...
` ` ▾

| **Apply** | **Close & Solve** | **Close** |

Data Items

Equal To - Specifies the new value for the selected *Set* item.

Set - Specifies the attribute data item to be changed.

Command Buttons

Apply - Applies the selected changes or executes the selected GDI Command.

Close - Closes the screen.

Close & Solve - Closes the screen, saves changes, and executes the Solution routine.

Notes & Considerations

● Multiple changes may be made to the selected features by changing the *Set* and *Equal To* items and repeatedly clicking the *Apply* command button.

● If a change is inadvertently made, and the *Allow Undo Of Data/Graphic Changes* preference settings option has been selected, immediately click the *Undo* icon to restore the original configuration.

● Features may be selected by individually clicking on each feature, by drawing a window around the desired features, by drawing a polygon around the desired features, or by a combination of all.

• To draw a selection window, hold down the *W* key, place the cursor at the desired location of the first corner, then left-click the mouse. Move the cursor to the other corner, and a dashed box will be displayed; left-click at the selected corner location. Only features completely within the selection window will be included in the selection set. If a feature already selected is selected again, it will be removed from the selection set.

• To draw a selection polygon, hold down the *P* key, place the cursor at the desired location of the first corner, then left-click the mouse. Move the cursor to the next point on the polygon. Continue until the next to last point is entered; press the *C* key to close the polygon. Only features which fall completely within the selection polygon will be included in the selection set. If a feature already selected is selected again, it will be removed from the selection set.

• To select an individual feature, left-click near the feature. If the feature already selected is selected again, it will be removed from the selection set.

• Press the *Enter* key to end the selection process.

● In addition to being able to change or set specific values, several GDI Commands may also be executed from the Multiple Edit Specifications screen. The available GDI Commands are listed in the following table:

Customer Features	
Command	*Description*
Assign Service Lines For Selected Customers	Assigns the selected customers to the geographically nearest supply main/pipe.
Clear Flags For Selected Customers	Clears flags assigned to the selected customers.
Delete Selected Customers	Deletes the selected customers.
Move Selected Customers	Moves the selected customers to a User-specified location. The relative configuration of the selected customers will not be changed.
Reassign Selected Customers Supply Main	Reassigns the selected customers to a new User-specified main/pipe.
Stack Selected Customers	"Stacks" the selected customers directly upon each other at a User-specified location.
Summarize Selected Customers	Displays a summary of the various data values of the selected customers.

Customer Features	
Command	***Description***
Unassign Selected Customers	Unassigns (unattaches) the selected customers from their current supply main(s).

Nodes	
Command	***Description***
Clear Flags For Selected Nodes	Executes the *Flag Unbroken Intersections* command for the selected nodes.
Clear Unbroken Intersection Flags	Removes (clears) displayed unbroken intersection flags assigned to the selected nodes.
Flag Unbroken Intersections	Executes the *Flag Unbroken Intersections* command for the selected nodes.
Move Selected Nodes	Moves the selected nodes to a User-specified location. The relative configuration of the selected nodes will not be changed.
Renumber Selected Nodes	Renumbers the selected nodes.
Report Selected Nodes	Displays the Node Data Report, containing only the selected nodes.
Summarize Selected Nodes	Displays a summary of the various data values associated with the selected nodes.

Pipe Features	
Commands	***Description***
Clear Flags For Selected Pipes	Automatically clears flags assigned to the selected pipes.
Convert Selected Pipes To Service Lines	Converts the selected pipes to customer service lines.
Copy Selected Pipes	Copies the selected pipes, including the associated nodes and optionally the attached customers, to a new User-specified location.
Delete Selected Pipes	Deletes the selected pipes.

Pipe Features	
Commands	*Description*
Export Selected Pipes To DXF	Exports the selected pipes into a DXF file. Executing this command will display the DXF Export Settings screen.
Extract Selected Pipes	Creates a new model from the selected pipes, including the associated nodes and, optionally, the assigned customers.
Join Selected Pipes	Joins the selected pipes, removing any associated nodes.
Move Selected Pipes	Moves the selected pipes to a User-specified location. The relative configuration of the selected pipes will not be changed.

User Graphic Images	
Commands	*Description*
Copy Selected User Graphic Images	Copies the selected User Graphic Images to a new User-specified location.
Delete Selected User Graphic Images	Deletes the selected User Graphic Images.
Move Selected User Graphic Images	Moves the selected User Graphic Images to a User-specified location. The relative configuration of the selected images will not be changed.
Stack Selected User Graphic Images	"Stacks" the selected User Graphic Images directly upon each other at a User-specified location.

User Graphic Lines	
Commands	*Description*
Copy Selected User Graphic Lines	Copies the selected User Graphic Lines to a new User-specified location.
Delete Selected User Graphic Lines	Deletes the selected User Graphic Lines.
Move Selected User Graphic Lines	Moves the selected User Graphic Lines to a User-specified location. The relative configuration of the selected lines will not be changed.
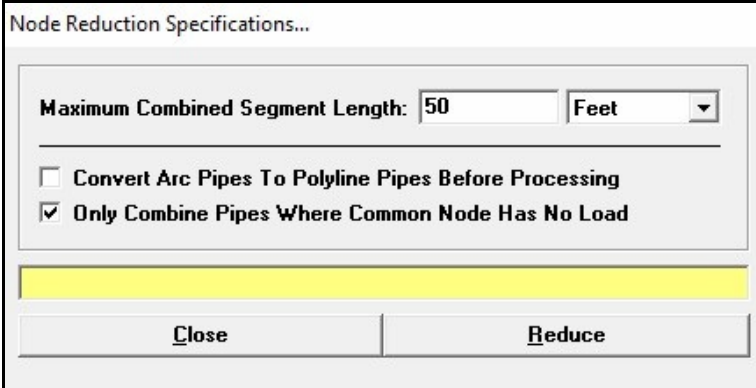
User Graphic Lines	
Commands	*Description*
Ungroup Selected User Graphic Lines	Ungroups the selected User Graphic Lines.

User Graphic Symbols	
Commands	*Description*
Copy Selected User Graphic Symbols	Copies the selected User Graphic Symbols to a new User-specified location.
Delete Selected User Graphic Symbols	Deletes the selected User Graphic Symbols.
Move Selected User Graphic Symbols	Moves the selected User Graphic Symbols to a User-specified location. The relative configuration of the selected symbols will not be changed.

User Text	
Commands	*Description*
Copy Selected User Text	Copies the selected User Text features to a new User-specified location.
Delete Selected User Text	Deletes the selected User Text features.
Move Selected User Text	Moves the selected User Text features to a User-specified location. The relative configuration of the selected text will not be changed.

GASWorkS™ 10.1   *GDI*

Node Reduction Specifications Screen

When the *Reduce Nodes* command is executed, the Node Reduction Specifications screen is displayed. The screen provides access to various settings and values used by the Reduction routine. The features associated with the screen are described as follows.

Displayed Items

Status Bar - Displays the status and results of the Reduce Nodes routine and the number of nodes eliminated.

Data Items

Convert Arc Pipes To Polyline Pipes Before Processing - When selected, arc-type pipe features will be converted to polyline-type pipe features before reducing the nodes in the model.

Maximum Combined Segment Length - Specifies the maximum length of the combined segments.

Only Combine Pipes Where Common Node Has No Load - When selected, pipe features are only combined when the common node has no load value.

Command Buttons

Close - Closes the screen.

Reduce - Proceeds with the Reduce Nodes routine.

Notes & Considerations

● The *Reduce Nodes* command is useful when a model has an unusually large number of nodes along similar-sized pipes. For example, when pipe segments have been imported from GIS data, and the segments were broken at each service tap location. Using the command will combine pipes of similar Size/Type codes where the combined length will be less than the specified Maximum Combined Segment Length value.

● If the results are unexpected, the original model configuration may be restored using the *Undo* command.

● Sometimes, it is beneficial to execute the command multiple times until no nodes are reported to have been eliminated.

Pipe Combination Specifications Screen

When the *Combine Pipes* command is executed, the Pipe Combination Specifications screen is displayed. The screen provides access to various settings and values used by the Combination routine. The features associated with the screen are described as follows.
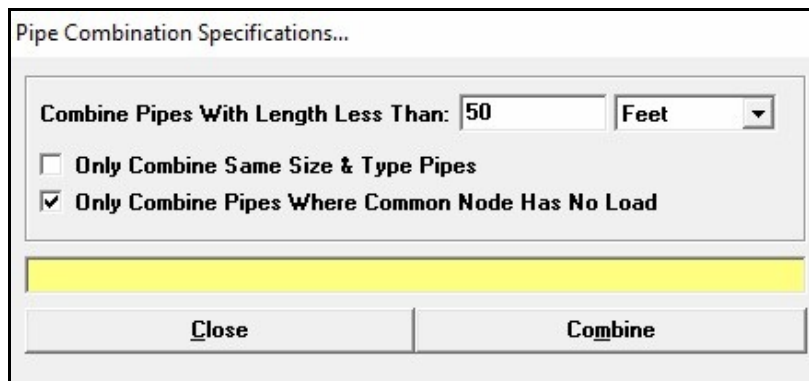
Displayed Items

Status - Displays the results and status of the Combine Pipes routine.

Data Items

Combine Pipes With Length Less Than - Specifies the maximum length value of a pipe segment to consider for combination.

Only Combine Pipes Where Common Node Has No Load - When selected, only pipes where the common node has node load will be combined.

Only Combine Same Size & Type Pipes - When selected, only pipes of the same size and type will be combined.

Command Buttons

Combine - Proceeds with the Combine Pipes routine.

Close - Closes the screen.

Notes & Considerations

● The *Combine Pipes* command is useful when a model has an unusually large number of short pipe segments. For example, when pipe segments have been imported from GIS data, and a short segment was included between a valve on a lateral and the header main that it connects to. Using the command will combine pipes where the length is less than the specified "Less Than" value.

● If the results are unexpected, the original model configuration may be restored using the *Undo* command.

Plot Preview

When the *Plot* command button is clicked on the Plot Settings screen, the Plot Preview will be displayed to allow the User to view the plot results before they are sent to the selected printer. The features associated with the screen are described as follows.

Displayed Items

Plot Preview - Displays a preview of the Plot image.

Command Buttons

Cancel - Closes the screen without plotting an image.

Continue - Proceeds with the Plot routine.

Notes & Considerations
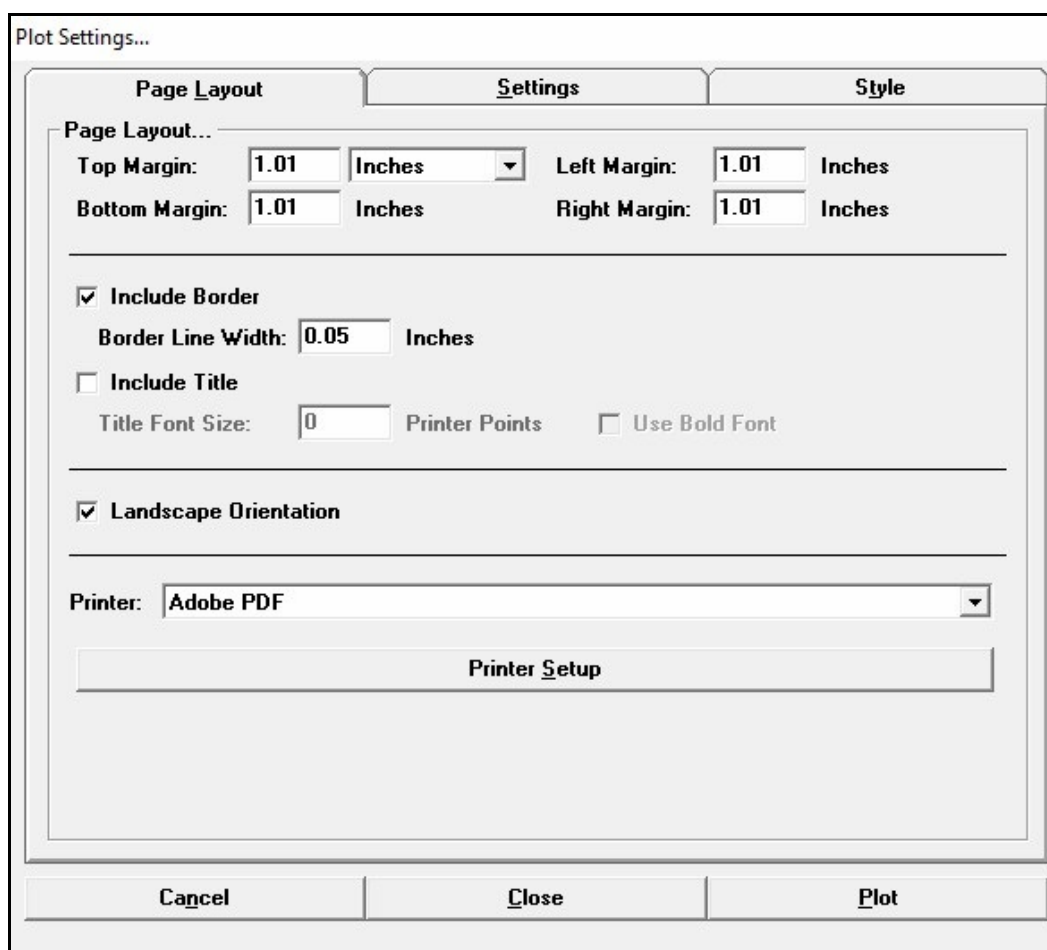
● After the plot image is generated before it is sent to the selected plotter/printer, it is displayed in the Plot Preview. Although a reduced image is displayed, the purpose of the preview is to allow the User to review the image and make adjustments before the image is sent to the plotter/printer. When displayed, review the previewed image; if the image looks acceptable, click the *Continue* command button to plot the image; otherwise, click the *Cancel* command button to cancel the process and revise the current settings.

Plot Settings Screen

When the *Plot GDI Image* command is executed, the Plot Settings screen is displayed to allow the User to set the various plot control settings. The features associated with the screen are described as follows.

Page Layout Tab

Data Items

Border Line Width - Specifies the line width used to draw the border. This item is only enabled when the *Include Border* option is selected.

Bottom Margin - Specifies the distance from the bottom of the paper page to the bottom of the plotted image.

Include Border - When selected, a border will be included with the plotted image.

Include Title - When selected, a title will be included with the plotted image.

Landscape Orientation - When selected, landscape orientation will be used when plotting. When not selected, the pages will be printed in portrait orientation.

Left Margin - Specifies the distance from the left side of the paper page to the left side of the plotted page.

Printer - Specifies the printer or plotter to be used. Select a printer from the list.

Right Margin - Specifies the distance from the right side of the paper page to the right side of the plotted image.

Title Font Size - Specifies the font size to be used when plotting. The size is expressed in printer points. One printer point is approximately equal to 1/72 of an inch. This item is only enabled when the *Include Title* option is selected.

Top Margin - Specifies the distance from the top of the paper page to the top of the plotted image.

Use Bold Font - When selected, the title text will be plotted using a bold (heavy) font style. This item is only enabled when the *Include Title* option is selected.

GASWorkS™ 10.1   *GDI*

Settings Tab

Data Items

Customer Symbol Size - Specifies the size of the customer symbols used to plot the image.

Fit To Page - When selected, the plotted image will be scaled to fit within the available plot space.

Font Name - Specifies the font name for plotting the associated data values. To change the Font Name, double-click in the field to open the Font settings screen. Select the desired name, then click the *OK* command button.

Font Size - Specifies the font size for plotting the associated data values.

Include Background Image - When selected, the attached background images will be included with the plotted image.

Node Symbol Size - Specifies the size of the node symbols used to plot the image.

Number Of Copies - Specifies the number of copies to be plotted.

Pipe Line Thickness - Specifies the thickness (width) of the lines used to plot the pipe-type features.

Plot Background Image Only - When selected, the attached background images will be the only items plotted.

Plot Customer Data Values - When selected, the data text items for the customer features will be included with the plotted image.

Plot In Black And White - When selected, the image will be plotted in black and white only. When not selected, the image will be plotted using the currently displayed screen colors. When a raster-type background image file is attached, the image may be printed in grayscale (not true black and white).

Plot Node Data Values - When selected, the data text items for the nodes will be included with the plotted image.

Plot Pipe Data Values - When selected, the data text items for the pipe features will be included with the plotted image.

Scale - Specifies the scale to plot the image. The scale value is expressed in terms of the currently selected printer and the *Coordinates* units. This item is only displayed if the *Fit To Page* item is not selected.

Use Bold Font - When selected, the data text items for the associated node, pipe, or customer features will be plotted using a bold (heavy) font style.

Style Tab
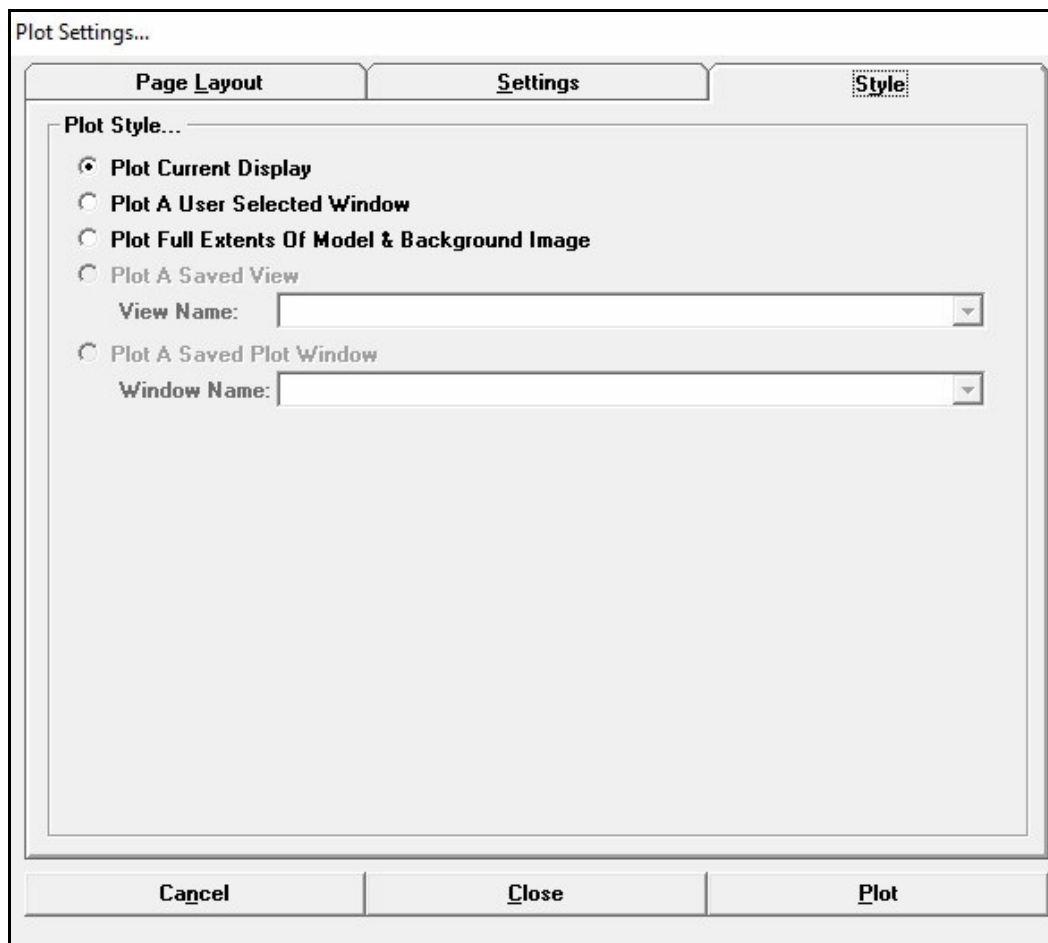
Data Items

Plot Current Display - Specifies that only the features of the model image currently displayed in the GDI Display will be plotted.

Plot Full Extents Of Model & Background Image - Specifies that all features associated with the model and the background image will be plotted.

Plot A Saved Plot Window - Specifies that the features displayed in a previously saved Plot Window will be plotted. If selected, then select the desired Plot Window from the associated *Window Name* list. This item is only enabled if a Plot Window was previously saved.

GASWorkS™ 10.1 *GDI*

Plot A Saved View - Specifies that the features displayed in a previously saved view will be plotted. If selected, then select the desired saved view from the associated *View Name* list.

Plot A User Selected Window - Specifies that the features displayed in a User-selected window will be plotted. If selected during the Plot routine, additional GDI Prompts will be displayed to allow the User to select the Plot Window.

View Name - Specifies the name of the saved view to plot.

Window Name - Specifies the name of the saved Plot Window to plot. This item is only enabled if a Plot Window was previously saved.

Command Buttons

Cancel - Closes the screen without saving changes or proceeding with the Plot routine.

Close - Closes the screen and saves changes.

Plot - Proceeds with the Plot routine.

Printer Setup - Displays the Print Setup screen. The screen's contents will vary depending on the selected printer and the current operating system. This command button is displayed on the *Page Layout* data tab.

Notes & Considerations
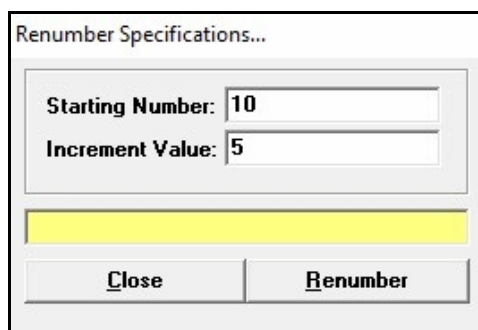
● Many printers and plotters have an internally set margin around the edges of the page. If the current margin values are less than these preset values, a message will be displayed when trying to plot. If this occurs, reset the margins to appropriate values.

● The selected printer is used to plot the image. The image is scaled as specified and centered within the margins of the currently selected page size and orientation. The page size may be changed by clicking the *Printer Setup* command button.

● The contents may be printed in a "PDF" format if a third-party PDF printer is installed. To print to a PDF file, select the PDF printer name from the *Printer* list, then click the *Print* command button.

● Getting the correct settings to create an acceptable plotted image is sometimes a trial and error-process. The image may need to be plotted a few times, adjusting settings between plots until the desired image results.

● The *Pipe Line Thickness*, *Node Symbol Size*, and *Customer Symbol Size* values are used as the base value for plotting the associated features. This means that this size value is used to plot all features whose size value is set to the "default" size value. Features with other size values will be plotted proportional to the base size value. For example, if the default Pipe Line Thickness is "2," all pipes with a display width set at "2" or to "default" will be plotted at the specified Pipe Line Thickness. If a pipe feature has a display width set at 1, it will be plotted at 1 divided by 2 (or 0.5) times smaller than the specified Pipe Line Thickness. And if a pipe feature has a display width set at 3, it will be plotted at 3 divided by 2 (or 1.5) times larger than the specified Pipe Line Thickness.

Renumber Specifications Screen

A routine is provided that allows the node Names (numbers) to be renumbered based on a User-specified Starting Number and Increment Value. The routine is controlled using the Renumber Specifications screen. The screen is accessed by executing the *Renumber Nodes* command. The features associated with the screen are described as follows.

Displayed Items

Status Bar - Displays the status of the Renumber Nodes routine.

Data Items

Increment Value - Specifies the increment value between the node numbers.

Starting Number - Specifies the value to start the numbering from.

Command Buttons

Close - Closes the screen.

Renumber - Proceeds with the Renumber Nodes routine.

Notes & Considerations

● The Renumber Nodes routine is performed as follows - The starting node number is assigned to the first node found in the node data file. The *Starting Number* is increased by the *Increment Value*, and this number is assigned to the next node in the file. This process is continued until all of the nodes have been renumbered.

Route Profile

The results of the *Show Route Profile* command are displayed on the Route Profile screen. The features associated with the screen are described as follows.
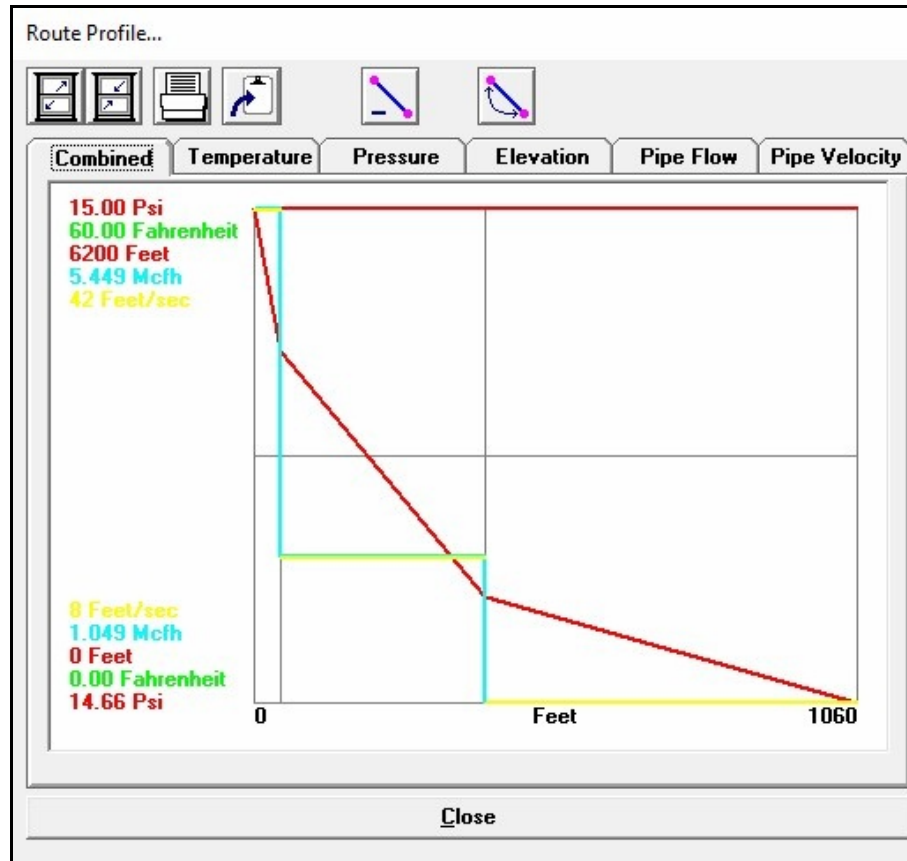
Icons

 Copy To Clipboard - Copies the contents of the Route Profile to the Windows Clipboard. Once copied, the contents may be pasted into another application.

 Delete Pipe - Deletes the selected (highlighted) pipe record.
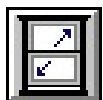
 Maximize Screen Size - Increases the Route Profile's size to fill the GASWorkS display.

Print - Prints the contents of the displayed data tab. When clicked, the Print Settings screen is displayed.
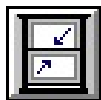
Reset Screen Size - Sets the Route Profile's size to its "normal" dimensions and position.

Swap Pipe Ends - Swaps (reorients) the From Node and To Node for the selected (highlighted) pipe segment in the Route Profile.

Command Buttons

Close - Closes the screen.

Notes & Considerations

● When the pipe segments are displayed in the Route Profile, they are oriented so that the flow is from left to right. That is, the pipe segments are oriented so that the upstream end is on the left and the downstream end is on the right. When manually selecting individual pipes for a route, it is generally best to start by selecting the most upstream segment and then sequentially select segments working downstream from there. The pipes are displayed in the order that they are selected. If a pipe is inadvertently selected, it may be unselected by clicking on it a second time. Only the highlighted pipes will be included in the Route Profile. If needed, the orientation may be changed using the *Swap Pipe Ends* command button.

● Depending on the selected Route Profile Options, some data tabs may not be enabled.

● The horizontal scale of the Route Profile represents the hydraulic distance along the route. The vertical scale of the Route Profile is automatically assigned based on the value range of the associated data item.

● The vertical lines on the profile represent node locations in the route.

● To display information about a pipe segment, click midway on the desired profile pipe segment (between vertical lines). A message will be displayed stating information about the selected pipe segment.

● To display information about a node, hover the mouse near the vertical line representing the desired node location. A message will be displayed stating information about the selected pipe segment.

● See the *Route Profile Options Screen* topic in the *Graphic Data Interface (GDI)* section for information about the *Show Route Profile* command and the Route Profile routine.

GASWorkS™ 10.1 *GDI*

Route Profile Options Screen

When the *Show Route Profile* command is executed, the Route Profile Options screen is displayed. The screen allows the various options associated with creating and displaying the Route Profile to be set by the User. The features associated with the screen are described as follows.
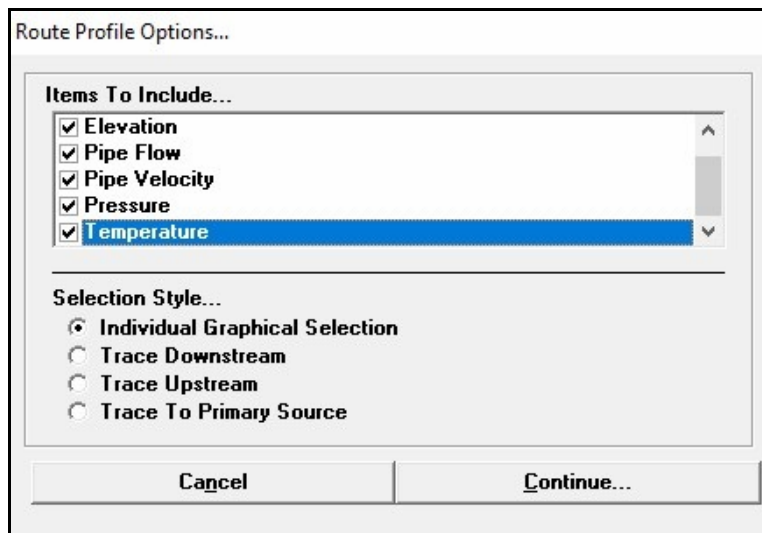
Data Items

Items To Include -

> **Combined Profile (Combines All Selected Items) -** When selected, all of the selected items will be combined in a single profile.
>
> **Elevation -** When selected, the elevation values will be included in the Route Profile.
>
> **Pipe Flow -** When selected, the pipe flow values will be included in the Route Profile.
>
> **Pipe Velocity -** When selected, the pipe velocity values will be included in the Route Profile.
>
> **Pressure -** When selected, the pressure values will be included in the Route Profile.
>
> **Temperature -** When selected, the temperature values will be included in the Route Profile.

Selection Style -

Individual Graphical Selection - Specifies that the pipe segments in the Route Profile will be selected individually.

Trace Downstream - Specifies that the pipe segments in the Route Profile will follow along the primary flow path downstream of the selected node location.

Trace To Primary Source - Specifies that the pipe segments in the Route Profile will follow along the primary flow path upstream of the selected node location until a regulator, compressor, or supply node is reached.

Trace Upstream - Specifies that the pipe segments in the Route Profile will follow along the primary flow path upstream of the selected node location.

Command Buttons

Cancel - Closes the screen without proceeding with the Route Profile routine.

Continue - Proceeds with the Route Profile routine. Depending on the route *Selection Style*, various prompts will be displayed to create the Route Profile.

Notes & Considerations

● When the *Combined Profile (Combines All Selected Items)* option is selected, the *Combination* data tab will contain a set of profiles representing the selected data item values.

● When using one of the trace methods to specify the Route Profile, the pipes found along the determined route will be highlighted, and a prompt will be displayed before the Route Profile is created. If the prompt appears, select the appropriate response to continue.

Shift Specifications Screen

The *Shift Model Coordinates* command allows a model to be graphically shifted, scaled, or rotated. The command is controlled using the Shift Specifications screen. The features associated with the screen are described as follows.

Displayed Items

Status - Displays the status of the Shift Coordinates routine.

Data Items

Rotation Angle - Specifies the angle to rotate the model before it is shifted. The rotation angle is measured as degrees clockwise from 12 o'clock (North). In terms of angle measurement, North is at the top of the display.

Rotation Point X - Specifies the east-west location of the pivot point for rotating the model. The value is expressed in *Coordinate* units.

Rotation Point Y - Specifies the north-south location of the pivot point for rotating the model. The value is expressed in *Coordinate* units.

Scale User Text & User Graphics - When selected, the User Text and User Graphic features will be scaled during the routine.

Update Pipe Lengths - When selected, the length values of the pipe features will be recalculated based on the new XY-coordinate values of the nodes and vertices. Do <u>not</u> select this option if the original length values were entered manually and the original values are to be maintained.

X Scale - Specifies the amount to increase or decrease the "X" coordinate values. Values less than one (1) will decrease the image size, and values greater than one (1) will increase the image size. The value is expressed as a multiplying factor, not as a percentage.

X Shift - Specifies the amount to move the model in the "X" (east-west) direction. Positive values move the model east, and negative values move the model west. The value is expressed in *Coordinate* units.

Y Scale - Specifies the amount to increase or decrease the "Y" coordinate values. Values less than one (1) will decrease the image size, and values greater than one (1) will increase the image size. The value is expressed as a multiplying factor, not as a percentage.

Y Shift - Specifies the amount to move the model in the "Y" (north-south) direction. Positive values move the model north, and negative values move the model south. The value is expressed in *Coordinate* units.

Z Scale - Specifies the amount to increase or decrease all coordinate values. Values less than one (1) will decrease the image size, and values greater than one (1) will increase the image size. The value is expressed as a multiplying factor, not as a percentage.

Z Shift - Specifies the amount to move the model in the vertical direction. Positive values move the model up, and negative values move the model down in "elevation." The value is expressed in *Coordinate* units.

Command Buttons

Close - Closes the screen.

Shift - Calculates the new coordinate values.

Notes & Considerations

● The new coordinate values are calculated in the following order. First, the model is rotated about the specified rotation location. Second, the Scale Factor is then applied. Last, the model is shifted by the specified amounts.

● This routine may move or rescale an imported model using different or incorrect scale or coordinate units.

● If the *Scale Text & User Graphics* option is selected, the User Text and User Graphics items are scaled and rotated by the entered scale and rotation values.

● This routine may be used to recalculate corrupt pipe length values by setting the "Shift" values to zero (0), the "Rotation" values to zero (0), and the "Scale" values to one (1). Ensure that the *Update Pipe Lengths* option is selected. When the shift is performed, the model will not move, however all of the length values will be recalculated based on the current node and vertex coordinate values.

Summary Of Selected Features

The Summary Of Selected Features is accessed by executing one of the "*Edit Multiple*" commands. From the Multiple Edit Specifications screen, on the *Commands* data tab, select the *Summarize Selected...* item, then click the *Apply* command button. The associated summary will be displayed. The features associated with the screen are described as follows.

```
Summary Of 5 Selected Customers...

  Total Number Of Units = 5

  Total Assigned Customer Loads (Volumetric) = -3000.000 Ccf/mo
  Total Adjusted Customer Loads (Volumetric) = -6000.000 Ccf/mo

  Total Customers Assigned To From Node = 0
  Total Customers Assigned To To Node = 0
  Total Customers Assigned To Both Nodes = 5
  Total Customers Assigned As Diversified = 0

  Total Customers Turned 'Off' = 0

  High Non-Zero Customer Load (Volumetric) = -600.000 Ccf/mo
  Low Non-Zero Customer Load (Volumetric) = -600.000 Ccf/mo

      [ Print ]                    [ Close ]
```

Displayed Items

Summary - Displays the associated information for the selected features.

Command Buttons

Close - Closes the screen.

Print - Prints the contents of the summary. When clicked, the Print Settings screen is displayed.

Notes & Considerations

● The "Summarize" command is useful when a total or summary of data values is required for a group of data features.

GASWorkS™ 10.1   *GDI*

Text Display Settings Screen

The Text Display Settings screen is displayed when the *Text Display Settings* command is executed or the *Text Display Settings* icon is clicked from the *Display Controls Toolbar*. The screen allows the text display colors and options to be set by the User. The features associated with the screen are described as follows.

Pipe Items Tab

Data Items

Bold Font - When selected, all pipe text items will be displayed using a bold (heavy) font style.

Dimensional Units - When selected, the dimensional units will be displayed as a suffix to the associated pipe text item, where appropriate.

Display Limit - Specifies the limit used to determine when to include or suppress the display of the pipe text items. To set the Display Limit to the current zoom level, double-click in the data field. When the diagonal distance across the GDI Display is less than the Display Limit, the pipe text items are displayed. The pipe text items are not displayed when the diagonal distance is greater than the Display Limit. The value is expressed in *Coordinate* units.

Display For Pipe Elements Only - When selected, the pipe text items are only displayed for the pipe elements (features).

Display Pipe Text Items - When selected, the pipe text items will be displayed in the model.

Items - When selected, indicates which pipe text items (data values) will be displayed.

Leader Lines On Moved Text - When selected, leader lines will be displayed when a pipe text item is moved.

Text Color - Specifies the color for displaying the pipe text items. To set a Color, click in the associated box. The Color Palette will be displayed. Select the desired color, then click the *OK* command button to assign the selected color.

Text Size - Specifies the text size for displaying the pipe data values. The value is expressed in the selected *Text Size Style* dimensional units.

Text Size Style - Specifies the pipe text size style (units).

Transparent Font - When selected, the text background will be displayed transparently. When not selected, the text will be displayed with an opaque background (the model image behind the text will be "blanked out").

Variable Identifiers - When selected, the item identifiers will be displayed as a prefix to each pipe text item. For example, a "Q =" prefix would be included when the pipe *Flow Rate* values are displayed.

Node Items Tab

Text Display Settings...

Pipe Items	Node Items	Customer Items

General Specifications...

☑ **Display Node Text Items**

☑ **Bold Font**
☑ Dimensional Units
☑ Leader Lines On Moved Text
☐ Only Display For Nodes With Non-Zero Loads
☐ Transparent Font
☐ Variable Identifiers

Text Size: `9`

Text Size Style: `Printer Points ▾`

Text Rotation: `90`

Display Limit: `2000`

Text Color: 🟥

Items...

☑ Node Name
☐ Node Atmospheric Pressure
☐ Node Coordinate Values (X,Y)
☐ Node Elevation
☐ Node Heating Value
☐ Node Load, Base
☐ Node Load, External
☐ Node Load, Total Adjusted
☐ Node Load, Total Unadjusted

Cancel	Apply

Data Items

Bold Font - When selected, all node text items will be displayed using a bold (heavy) font style.

Dimensional Units - When selected, the dimensional units are displayed as a suffix to the associated node text item, where appropriate.

Display Limit - Specifies the limit used to determine when to include or suppress the display of the node text items. To set the Display Limit to the current zoom level, double-click in the data field. When the diagonal distance across the GDI Display is less than the Display Limit, the pipe text items are displayed. The pipe text items are not displayed when the diagonal distance is greater than the Display Limit. The value is expressed in *Coordinate* units.

Display Node Text Items - When selected, the node text items will be displayed in the model.

Items - When selected, indicates which node text items (data values) will be displayed.

Leader Lines On Moved Text - When selected, leader lines will be displayed when a node text item is moved.

Only Display For Nodes With Non-Zero Loads - When selected, text items are only displayed for nodes with a non-zero load.

Text Color - Specifies the color for displaying the node text items. To set a Color, click in the associated box. The Color Palette will be displayed. Select the desired color, then click the *OK* command button to assign the selected color.

Text Rotation - Specifies the rotation angle for displaying the node text items. The rotation angle is measured as degrees clockwise from 12 o'clock (North). In terms of angle measurement, North is at the top of the display.

Text Size - Specifies the text size for displaying the node text items. The value is expressed in the selected *Text Size Style* dimensional units.

Text Size Style - Specifies the node text size style (units).

Transparent Font - When selected, the text background will be displayed transparently. When not selected, the text will be displayed with an opaque background (the model image behind the text will be "blanked out").

Variable Identifiers - When selected, the item identifiers will be displayed as a prefix to each node text item. For example, a "P =" prefix would be included when the *Pressure* values are displayed.

GASWorkS™ 10.1   *GDI*

Customer Items Tab

Data Items

Bold Font - When selected, all customer text items will be displayed using a bold (heavy) font style.

Dimensional Units - When selected, the dimensional units are displayed as a suffix to the associated customer text item, where appropriate.

Display Customer Text Items - When selected, the customer text items will be displayed in the model.

Display Limit - Specifies the limit used to determine when to include or suppress the display of the customer text items. To set the Display Limit to the current zoom level, double-click in the data field. When the diagonal distance across the GDI Display is less than the Display Limit, the pipe text items are displayed. The pipe text items are not displayed when the diagonal distance is greater than the Display Limit. The value is expressed in *Coordinate* units.

Items - When selected, the customer text items (data values) will be displayed.

Leader Lines On Moved Text - When selected, leader lines will be displayed when a customer text item is moved.

Text Color - Specifies the color for displaying the customer text items. To set a Color, click in the associated box. The Color Palette will be displayed. Select the desired color, then click the *OK* command button to assign the selected color.

Text Rotation - Specifies the rotation angle for displaying the customer text items. The rotation angle is measured as degrees clockwise from 12 o'clock (North). In terms of angle measurement, North is at the top of the display.

Text Size - Specifies text size for displaying the customer text items. The value is expressed in the selected *Text Size Style* dimensional units.

Text Size Style - Specifies the customer text size style (units).

Transparent Font - When selected, the text background will be displayed transparently. When not selected, the text will be displayed with an opaque background (the model image behind the text will be "blanked out").

Variable Identifiers - When selected, the item identifiers will be displayed as a prefix to each customer text item. For example, a "Q =" prefix would be included when *Customer Load* values are displayed.

Command Buttons

Apply - Closes the screen, saves changes, and applies the current settings to the displayed image.

Cancel - Closes the screen without saving or applying changes.

Notes & Considerations

● Display of the "data" text values for the various data features is controlled with the Text Display Settings screen. Individual settings are provided for pipe, node, and customer data features. The settings are applied uniquely to each of the GDI Display Panels. In other words, separate settings are saved for each Display Panel.

● The data text size may be set to a constant value (Printer Points) or a variable value (Coordinates). When set to a constant value, the displayed text size remains the same regardless of the "zoom" level. When set to a variable value, the displayed text size decreases and enlarges as the "zoom" level changes.

● The location of the displayed text may be moved for individual data features using the associated "*Move Text*" command or the associated grips.

● The text display may be turned on or off at certain zoom levels using the associated "Display Limit" values. When the diagonal distance across the GDI display is less than the Display Limit, the text is displayed. When the diagonal distance is greater than the Display Limit, the text display is suppressed. Use a value of zero (0) to cause the text to always be displayed.

Trace & Update Data Specifications Screen

The Trace & Update Data routine is controlled using the Trace & Update Data Specifications screen. To access the screen, execute the *Trace & Update Data* command. The features associated with the screen are described as follows.

Data Items

Equal To - Specifies the new value for the selected *Set* item.

Highlight Customers - When selected, customers found along the trace route will be highlighted using the selected *Trace Highlight Color*.

Set - Specifies which data item is to be updated.

Start Trace At - Specifies the type of feature to use as the starting point for the trace.

Trace Highlight Color - Specifies the color to use when highlighting the features associated with the trace. To set a Color, click in the associated box. The Color Palette will be displayed. Select the desired color, then click the *OK* command button to assign the color.

Trace Style - Specifies the style of the trace to be created. See the following *Notes & Considerations* section for a description of each Trace Style.

Command Buttons

Cancel - Closes the screen without proceeding with the Trace routine.

Trace & Update - Closes the screen, performs the trace, and updates the specified item values.

Notes & Considerations

● All traces will stop at a turned "off" (*Status* is "off") pipe feature.

● There are several types of *Trace Styles*, and each behaves differently. See the *Trace Screen* topic in the *Graphic Data Interface (GDI)* section for information about the *Trace Styles*.

● Use the *Undo* command to restore changes made by this command.

Trace Results

When the *Create A Trace Results File* option is selected on the Trace Specifications screen, the results are stored in a file is called the Trace Results. The Trace Results may be viewed or printed by executing the *View Trace Results* command. The features associated with the screen are described as follows.
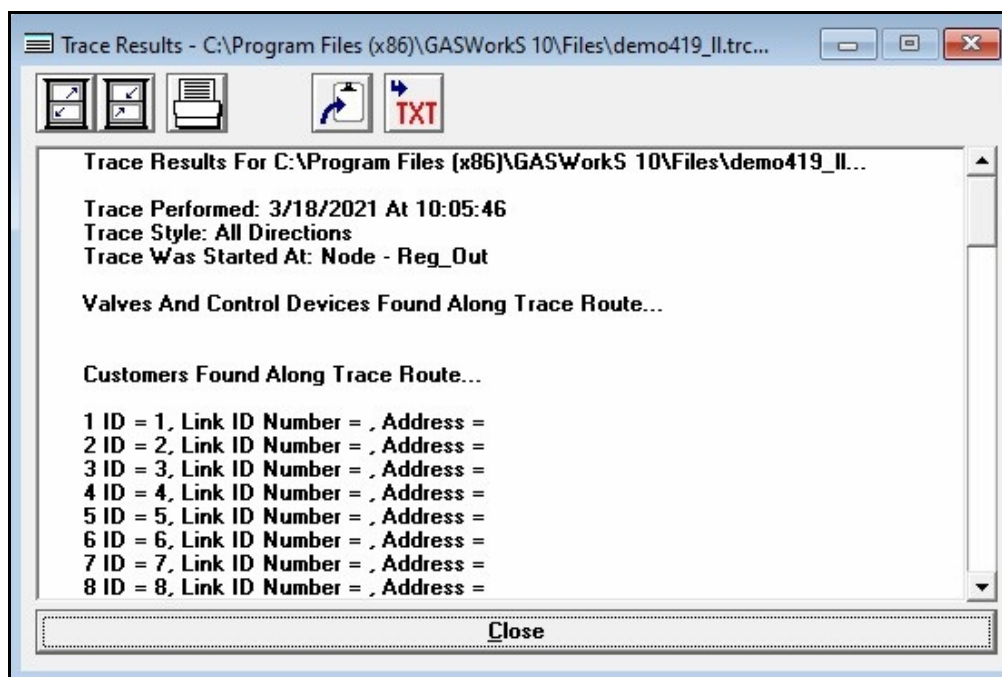
Displayed Items

Trace Results - Displays the Trace Results. To browse the display, adjust the vertical scroll bar adjacent to the display area.

Icons

Copy To Clipboard - Copies the contents of the Trace Results to the Windows Clipboard. Once copied, the contents may be pasted into another application.
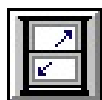
Maximize Screen Size - Increases the Trace Result's size to fill the GASWorkS display.

Print - Prints the contents of the Trace Results. When clicked, the Print Settings screen is displayed.

Reset Screen Size - Sets the Trace Result's size to its "normal" dimensions and position.

Save To Text File - Saves the data of the Trace Results contents to a text file (.txt). When clicked, the File Selection screen is displayed.

Command Buttons

Close - Closes the screen.

Notes & Considerations

● When the *Create A Trace Results File* option is selected, the findings of the trace routine are reported in the Trace Report.

● For customer addresses to be reported in the Trace Report, a Customer Attribute file with an assigned "Address" field must be attached.

Trace Specifications Screen

The Trace routine is controlled using the Trace Specifications screen. The screen is accessed by executing the *Trace* command or clicking the *Trace* icon from the *Utility Commands Toolbar*. The features associated with the screen are described as follows.
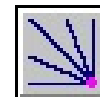
```
┌─ Trace Specifications... ──────────────────┐
│ ┌─ Trace Style... ───────────────────────┐ │
│ │ Trace All Directions               ▼  │ │
│ └────────────────────────────────────────┘ │
│ ┌─ Start Trace At... ────────────────────┐ │
│ │ ⦿ A Node     ○ A Pipe    ○ A Customer │ │
│ └────────────────────────────────────────┘ │
│ ┌─ Options... ───────────────────────────┐ │
│ │ ■  Trace Highlight Color               │ │
│ │ ☐  Highlight Customers                 │ │
│ │ ☐  Reset Feature Colors Before Tracing │ │
│ │ ☐  Blend Colors On Overlapping Trace   │ │
│ │    Paths                               │ │
│ │ ─────────────────────────────────────  │ │
│ │ ☑  Create A Trace Results File         │ │
│ │ ┌────────────────────────────────────┐ │ │
│ │ │☐Report Customers Along Route     ▲│ │ │
│ │ │☐Report Pipes Along Route          │ │ │
│ │ │☐Report Regulators & Set Pressures ▼│ │ │
│ │ │  Feeding Into Rout                 │ │ │
│ │ └────────────────────────────────────┘ │ │
│ └────────────────────────────────────────┘ │
│ ┌────────────────────────────────────────┐ │
│ │          Reset Feature Colors          │ │
│ └────────────────────────────────────────┘ │
│ ┌──────────────────┐ ┌───────────────────┐ │
│ │      Cancel       │ │      Trace        │ │
│ └──────────────────┘ └───────────────────┘ │
└─────────────────────────────────────────────┘
```

Data Items

Blend Colors On Overlapping Trace Paths - When selected, the overlapping trace path colors will be blended.

Create A Trace Results File - When selected, a Trace Results file will be created.

Highlight Customers - When selected, the customers found along the trace route will be highlighted with the selected *Trace Highlight Color*.

Report Customers Along Route - When selected, the customers found along the trace route will be included in the Trace Results file. This item is enabled when the *Create A Trace Results File* option is selected.

Report Pipes Along Route - When selected, the pipes found along the trace route will be included in the Trace Results file. This item is enabled when the *Create A Trace Results File* option is selected.

Report Regulators & Set Pressures Feeding Into Route - When selected, the regulators and set pressures found along the trace route will be included in the Trace Results file. This item is enabled when the *Create A Trace Results File* option is selected.

Report Summary Of Features Along Route - When selected, all of its features found along the trace route will be included in the Trace Results file. This item is enabled when the *Create A Trace Results File* option is selected.

Report Valves & Control Devices Along Route - When selected, found valve nodes, regulators, and compressors found along the trace route will be included in the Trace Results file. This item is enabled when the *Create A Trace Results File* option is selected.

Reset Feature Colors Before Tracing - When selected, the feature colors will be reset to their currently saved values before the trace is performed.

Start Trace At - Specifies the type of feature to use as the starting point for the trace.

Trace Highlight Color - Specifies the color to use when highlighting the features associated with the trace. To set a Color, click in the associated box. The Color Palette will be displayed. Select the desired color, then click the *OK* command button to assign the selected color.

Trace Style - Specifies the style of the trace to be created. See the following *Notes & Considerations* section for a description of each Trace Style.

Command Buttons

Cancel - Closes the screen without proceeding with the Trace routine.

Reset Feature Colors - Resets the feature colors to the current values.

Trace - Closes the screen and proceeds with the Trace routine.

Notes & Considerations

● If the *Create A Trace Results File* option is selected, the File Selection screen is displayed. Enter the desired name, then click the *Continue* command button. The Trace Results file can possess any valid name and does not necessarily need to be named similarly to the model.

● All traces will stop at a turned "off" (*Status* is "off") pipe feature.

● There are several types of *Trace Styles*, and each behaves differently.

Trace All Directions - Performs a trace in all directions, stopping only at pipes that are turned "off." All pipes which are connected to the specified starting feature will be highlighted.

Trace Downstream - All Flow Paths - Performs a downstream trace. All pipes fed (flow) downstream from the specified starting feature will be highlighted.

Trace Downstream - Primary Flow Path Only - Performs a downstream trace. The pipes with the highest flow rate fed (flow) downstream from the specified starting feature will be highlighted.

Trace To Any Flow Control Device - Performs a trace in all directions, stopping at valve nodes, regulators, compressors, or pipes that are turned "off." All pipes connected to the specified starting feature will be highlighted until a valve node, regulator, or compressor-type feature is encountered.

Trace To Maintained Valves Only - Performs a trace in all directions, stopping at maintained valve nodes or other flow control devices. All pipes connected to the specified starting feature will be highlighted until a maintained valve node is encountered.

Trace To Primary Source - Performs an upstream trace following the path with the highest flow rate, which feeds (flows) into the specified starting feature, stopping at a regulator, compressor, or supply node. All pipes found along this path will be highlighted.

Trace To Regulators - Performs a trace to all regulators that are hydraulically connected to the selected pipe(s), regardless of flow direction.

Trace Upstream - All Flow Paths - Performs an upstream trace. All pipes which feed (flow) upstream from the specified starting feature will be highlighted.

Trace Upstream - Primary Flow Path Only - Performs an upstream trace. The pipes with the highest flow rate, which feed (flow) from upstream into the specified starting feature, will be highlighted.

● See the *Valve Nodes* topic in the *Miscellaneous* section for a complete description of valve nodes, their usage, and their relationship to the Trace routines.

GASWorkS™ 10.1   *GDI*

User Table Editor

To edit the contents of a previously created User Table, click on a table feature and click the *Yes* command button on the displayed message or use the *Edit User Table* command. The User Table Editor screen is displayed. The features associated with the screen are described as follows.
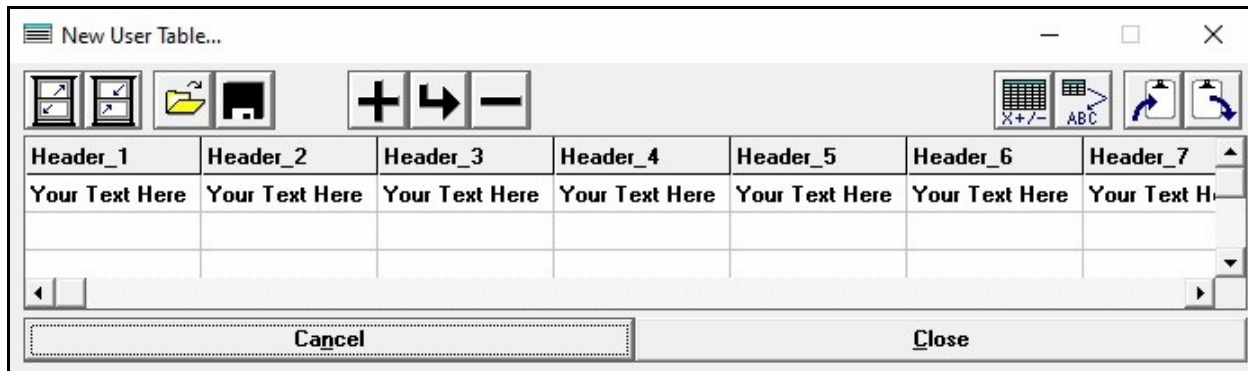
Data Items

User Table - A spreadsheet-style "report" containing the text values to be contained in the table.

Icons
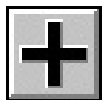
 Add Row - Adds a new row to the end of the table.

 Add As User Text - Inserts the associated content as User Text into the model image.

 Copy To Clipboard - Copies the contents of the User Table to the Windows Clipboard. Once copied, the contents may be pasted into another application.
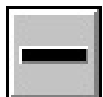
 Delete Row - Deletes the selected (highlighted) row.

GASWorkS™ 10.1

Insert From Clipboard - Inserts data from the Windows Clipboard. The data will be inserted into the current User Table immediately after the selected (highlighted) row. The data must be in a tab-delimited format that only contains "data" values and does not include header titles or record numbers. Generally, it is best to insert data that has been previously copied from a User Table using the *Copy To Clipboard* icon.

Insert Row - Inserts a new row just before the selected (highlighted) row.
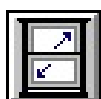
Maximize Screen Size - Increases the User Table's size to fill the GASWorkS display.

Open - Opens a previously saved User Table file (.csv). When clicked, the File Selection screen is displayed.
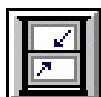
Reset Screen Size - Sets the User Table's size to its "normal" dimensions and position.
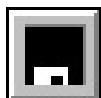
Save - Saves the contents of the currently displayed User Table. When clicked, the File Selection screen is displayed. Once saved, the file may be opened and used with other models.

View In Spreadsheet Application - Displays the User Table contents in the application currently assigned to the ".csv" file format in the default system application assigned to the ".csv" file type. When clicked, the assigned application will open and display the contents of the User Table.

Command Buttons

Cancel - Closes the screen without saving changes.

Close - Closes the User Table Editor.

Notes & Considerations

● The content of the table may be saved to a Comma-Separated Values file (.csv). Once saved, the file can opened and used in other models or applications.

● Appropriately foraged data may be inserted into a User table. To properly insert clipboard data into the User Table, the data must be in a tab-delimited format that only contains "data" values and does not include header titles or record numbers. Generally, it is best to insert data that has been previously copied from the User Table using the *Copy To Clipboard* icon. However, appropriately formatted data may be copied and inserted from any application.

GASWorkS™ 10.1

Data Panel

DATA PANEL

Customer Data & Results

If customer features are included in the model, the data associated with the customers may be accessed through one of the various *Edit Customer Data* methods. The Customer Data is displayed in the Data Panel. The features associated with the customer data are described in the following sections.
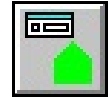
Customer Data & Results...	
Record Number	6
Internal ID Number	6
Supply Main Number	5

Displayed Items

Internal ID Number - Displays the customer's unique (to GASWorkS) internal identification number. This value is automatically assigned and cannot be changed by the User.

Record Number - Displays the position of the current record in the customer data file. This value is automatically assigned and cannot be changed by the User.

Supply Main Number - Displays the Link ID Number of the pipe (main) that supplies the customer (the pipe the customer is assigned to). The customer's load will be assigned to the external load of the pipe's From Node or To Node as specified by the *Node Load Application* setting. This value cannot be changed directly by the User. *Note -* If the assigned pipe does not have an assigned Link ID Number, the Record Number of the associated pipe will be displayed.

Hydraulic Data Items

[-] Hydraulic Data Items...	
Node Load Application	Both Nodes
Unit Count	1
Per Unit Load	-600
Load Units	Ccf/mo
Adjust Load	Yes
Load Status	On

GASWorkS™ 10.1 *Data Panel*

Displayed Items

Supplied By Customer **ID -** Only displayed if the customer is a "branch" customer. Displays the Internal ID Number of the trunk customer to which the branch customer is attached.

Supplies Customer ID - Only displayed if the customer is a "trunk" customer. Displays the Internal ID Number of the branch customer assigned to it.

Data Items

Adjust Load - Indicates whether the customer's *Per Unit Load* value is multiplied by the Design Factor when the load is applied to the associated node(s). The base load value is not changed by this setting.

Heating Type - Specifies the space heating type when "Diversified" is selected for the *Node Load Application*.

Load Status - Indicates whether the customer's *Per Unit Load* value is used ("on") or not used ("off") in the model if the status is "off," the applied external load value and the calculated *Per Unit Adjusted Load* value are equal to zero (0).

Load Units - Specifies the dimensional units for the *Per Unit Load* value.

Node Load Application - Specifies how to apply the customer's per unit load to the supply pipe and associated nodes.

Per Unit Load - Specifies the load value for each customer "unit." The item may be manually entered or read from an attribute file (database) if one is attached. A negative value indicates a demand (flow leaving the system), and a positive value indicates a supply (flow entering the system). The load value is expressed in the assigned *Load Units*.

Unit Count - Specifies the total number of units (meters or customers) the selected customer feature represents. The value must be in the range of 1 to 32,000.

Graphic Data Items

[-] Graphic Data Items...	
X-Coordinate	-104.815640
Y-Coordinate	38.853391
Tap X-Coordinate	-104.815425
Tap Y-Coordinate	38.853385
Coordinate Units	Web L/L
Symbol Style	Default
Symbol Size Style	Default
Symbol Color	Default
Selected Color	
Service Line Type	2-Point

Displayed Items

Coordinate Units - Displays the dimensional units used for the various coordinate values.

Service Line Type - Displays the graphic line type of the service line attached to the customer.

Tap X-Coordinate - Displays the east-west location of the service tap on the pipe (main) that the associated customer is assigned to.

Tap Y-Coordinate - Displays the north-south location of the service tap on the pipe (main) that the associated customer is assigned to.

Vertex Count - If the customer uses a polyline type service line, displays the number of vertices associated with the line.

Data Items

Selected Color - Specifies or displays the color of the customer symbol. To set a color, select "Selected Color" from the *Symbol Color* list, then click in the adjacent cell. The Color Palette will be displayed. Select the desired color, then click the *OK* command button to assign the selected color.

Symbol Color - Specifies whether the "Default" color or a "Selected Color" will be used to display the customer's symbol.

Symbol Size - If the *Symbol Size Style* is not set to "Default," specifies the size of the customer symbol.

Symbol Size Style - Specifies the style for the Customer symbol size. An "Absolute" style will cause the symbol to change in size as the model image zoom level changes. A "Relative" style will cause the symbol to be displayed at a consistent size as the zoom level changes.

Symbol Style - Specifies the style of the symbol used to display the customer locations.

X-Coordinate - Specifies the customer's east-west location.

Y-Coordinate - Specifies the customer's north-south location.

Text Display Items

[-] Text Display Items...	
Display Text Items	Yes
Text Location Style	Default

Displayed Items

Text Location Style - Displays the orientation of the customer text items.

Data Items

Display Text Items - Indicates whether the data text items for the customer will be displayed. The customer text items will only be displayed if the *Display Customer Items* option is selected in the Text Display Settings.

Text Rotation - If the *Text Location Style* is "User Selected," specifies the rotation angle of the customer text items.

Text X - If the *Text Location Style* is "User Selected," specifies the east-west location of the customer text items.

Text Y - If the *Text Location Style* is "User Selected," specifies the north-south location of the customer text items.

GASWorkS™ 10.1

Miscellaneous Data Values

[-] Miscellaneous Data Values...	
Pipe Facility Type	UnSpecified
Edit Date	12/24/2019

Displayed Items

Edit Date - Displays the date when the customer data was last changed.

Pipe Facility Type - Displays the Facility Type of the pipe (main) that the customer is attached to. The customer inherits the same Facility Type as the assigned pipe (main).

Attribute Data Values

If a customer attribute file (database) is attached to the model, the *Attribute Data Values* section will display the values contained in the associated database. The values in this section will vary. If a "linked" style database is attached, the values may be changed by the User. If a "non-linked" style database is attached, the values are view-only and cannot be changed by the User. The persistent contents of the list are described as follows.

[-] Attribute Data Values...	
Link ID Number	5
Link To Database	Yes

Data Items

Link To Database - Indicates whether the customer record should be linked to the attribute file (database). If "Yes" is selected, the customer will be linked to the database by matching the Link ID Number.

Link ID Number - Specifies the identification number used to link the customer feature in the model to the attribute file. Or, if no attribute file is attached, a User-defined identifier for the customer. When a "linked" style database is attached, the number is automatically assigned and cannot be changed by the User. When a "non-linked" style database is attached, the number is assigned by the User. The number should generally represent a unique value in the attribute file (database). When no database is attached, the number may be used to maintain any User desired alphanumeric value. The value may be a combination of up to twenty (20) alphanumeric characters.

GASWorkS™ 10.1

Data Panel

Calculated Values

[-] Calculated Values...	
Per Unit Adjusted Load	-78.000 Cfh
Total Adjusted Load	-312.000 Cfh

Displayed Items

Per Unit Adjusted Load - If the *Adjust Load* item is "Yes," displays the *Per Unit Load* value times the Design Factor. Otherwise, displays the *Per Unit Load* value.

Total Adjusted Load - Displays the *Unit Count* value times the *Per Unit Adjusted Load* value.

Actions

Displays the various GDI Commands that may be executed for the Customer Data Panel.

[-] Actions - Select One...
Add Associated User Text
Add Vertex To Selected Customer
Copy Selected Customer
Delete Selected Customer
Edit Multiple Customers
Flag Selected Customer
Match All Customer Properties
Match Customer Graphic Properties
Match Customer Hydraulic Properties
Move Selected Customer
Move Tap For Selected Customer
Print Data List Values
Reassign Supply Main For Selected Customer
Unassign Selected Customer From Supply Main

Action	*GDI Command* **- See Description**
Add Associated User Text	*Add User Text - Associated*

GASWorkS™ 10.1　　　*Data Panel*

Action	*GDI Command* - See Description
Add Vertex To Selected Customer	*Add Service Line Vertex*
Clear Flag For Selected Customer	*Clear Customer Flag*
Copy Selected Customer	*Copy Customer*
Delete Selected Customer	*Delete Customer*
Delete Vertex From Selected Customer	*Delete Service Line Vertex*
Edit Multiple Customers	*Edit Multiple Customers*
Flag Selected Customer	*Flag Customer*
Match All Customer Properties	*Match Customer Properties (All)*
Match Customer Graphic Properties	*Match Customer Graphic Properties*
Match Customer Hydraulic Properties	*Match Customer Hydraulic Properties*
Move Selected Customer	*Move Customer*
Move Tap For Selected Customer	*Move Service Tap*
Move Text For Selected Customer	*Move Customer Text*
Move Vertex For Selected Customer	*Move Service Line Vertex*
Print Data List Values	None - Prints the contents of the data list.
Reassign Supply Main For Selected Customer	*Reassign Customer Supply Main*
Unassign Selected Customer From Supply Main	None - Removes the supply main, leaving the customer unassigned.

Command Buttons

Apply Data Values - Saves and applies changes.

Continue - Proceed with the associated routine without making changes.

Dismiss - Closes the Data Panel if no changes were made.

Notes & Considerations

● Data cells with a "yellow" background color display assigned or calculated values. Values in these cells cannot be directly changed by the User.

● The *Node Load Application* options are described as follows:

Both Nodes - Indicates that one-half of the load will be applied to the *External Load* value of the assigned pipes From Node and To Node.

Diversified - Indicates that the load will be applied using the diversified load calculation method associated with the selected *Heating Type* method.

From Node - Indicates that the entire load will be applied to the *External Load* value of the assigned pipe's From Node.

None - Indicates that the load is not applied to either node.

To Node - Indicates that the entire load will be applied to the *External Load* value of the assigned pipe's To Node.

● If an attribute file (database) is attached, the Link ID Number matches the selected data feature and the corresponding record in the database. The first record in the database with a matching Link ID Number is displayed in the *Attribute Data Values* section. If a database is not attached, the Link ID Number may be used to maintain any desired value. The external value will not affect the model solution. See the *Working With An Attribute Database* topic in the *Miscellaneous* section for more information on using an attribute database. The value may be up to twenty (20) alphanumeric characters.

Favorite Results

The Favorite Results may be accessed by selecting the *Favorite Results* item from the Data Panel. The features associated with the Favorite Results are described in the following sections.

Displayed Items

Favorite Results - Displays a list of results for the nodes and pipes previously indicated as "favorites."

Actions

Displays the various routines that may be executed for the Favorite Results.

Action	*GDI Command* - **See Description**
Add A Favorite	None - Allows a node or pipe feature to be selected for inclusion in the *Favorite Results* list.
Print Favorites	None - Prints the contents of the Favorites List.

Notes & Considerations

● The Favorite Results list allows for a quick review of specific values for specific pipe or node features. If the *Display Favorite Results List Upon Completion* solution option is selected, the list will be automatically displayed after an executed solution is complete. To display the list at other times, select the *Favorite Results* option from the list at the top of the Data Panel.

● Several methods are available for adding features to the Favorite Results list; one is by using the "Add To Favorites Results" in the *Actions* section of the Data Panel when a pipe or node is selected. When a feature is being added, the Favorite Options screen is displayed. Select the appropriate item from the list and click *Continue* to add the item to the Favorite Results. To remove a feature from the Favorite Results, click in the (gray) cell with the features identifying information, press the *Delete* key, and respond to the prompt accordingly.

● To "zoom to" a feature in the Favorite Results and display its data and results, click in the (yellow) cell containing the item value. The data associated with the selected feature will be displayed in the Data Panel.

Fitting Selection Screen

The Fitting Selection screen may be used to attach (add) fittings to a selected pipe segment. This screen is displayed when the "Click To Add..." item is clicked from the *Fitting* cell when the Pipe Data is displayed in the Data Panel. The features associated with the screen are described as follows.
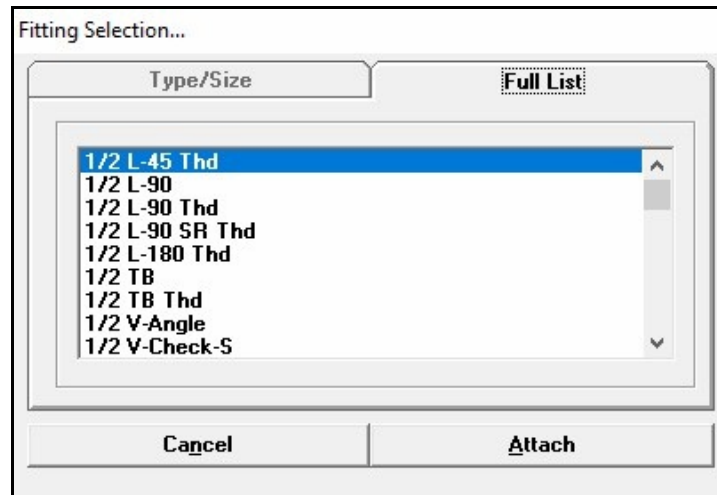
Displayed Items

Full List - Displays the available Size/Type Codes (fittings) in the current Fitting Property Table.

Command Buttons

Attach - Closes the screen and attaches the selected fitting.

Cancel - Closes the screen without attaching a fitting.

Notes & Considerations

● Only Size/Type Codes where the *Display* item in the Fitting Property Table is "Yes" are displayed in the *Full List*.

● Fittings may only be attached to pipe features where the Hydraulic Type is "Pipe."

Node Data & Results

The data associated with the nodes may be accessed using one of the various *Edit Node Data* methods. The Node Data is displayed in the Data Panel. The features associated with the node data are described in the following sections.

Node Data & Results...	
Name	10
Record Number	2
Internal ID Number	4
Pipe Node 1	Reg_Out
Pipe Node 2	15
Pipe Node 3	25

Displayed Items

Internal ID Number - Displays the node's unique (to GASWorkS) Internal ID Number.

Name - Displays the name or number of the associated node.

Pipe Node 1, 2, ... - Displays the node Name for the pipes connected to the selected node. Click in the adjacent cell to display the associated Pipe's Data.

Record Number - Displays the position of the current node record in the node data file. This value is automatically assigned and cannot be changed by the User.

Hydraulic Data Items

[-] Hydraulic Data Items...	
Name	10
Pressure	14.90347
Pressure Units	Psi
Pressure Known	No
Base Load	0
Load Units	Mcfh
Base Load Known	Yes
Adjust Base Load	Yes
Base Load Status	On
Adjust External Load	Yes
External Load Status	On
Elevation	6200
Elevation Units	Feet
Temperature	60
Temperature Units	Fahrenheit
Temperature Known	Yes
Hydraulic Type	Connection

Displayed Items

Elevation Units - Displays the dimensional units used for the *Elevation* value.

Temperature Units - Displays the dimensional units used for the *Temperature* value.

Data Items

Adjust Base Load - Indicates whether the node's base load value will be multiplied by the Design Factor during the Solution routine. The original data value is not changed by this setting; the *Base Load* value is only modified during the Solution routine.

Adjust External Load - Indicates whether the node's external load value will be multiplied by the Design Factor during the Solution routine. The original data value is not changed by this setting; the *External Load* value is only modified during the Solution routine.

Base Load - Specifies the node's base load value. If the model has been recently solved and the load is "unknown," the item will display the calculated load value for the node.

Base Load Known - Indicates whether the base load value is "known" ("Yes") or "unknown" ("No"). If set to unknown, the Base Load value will be calculated during the Solution routine.

Base Load Status - Indicates whether the base load value is "on" or "off" during the Solution routine. If the status is "off," the *Base Load* value is assumed to be zero (0) during the Solution routine. The original data value is not changed by this setting.

Elevation - Specifies the node's height above sea level. Initially, the field is populated with the default *Node Elevation* value.

External Load Status - Indicates whether the external load value is "on" or "off" during the Solution routine. If the status is "off," the *External Load* value is assumed to be zero (0) during the Solution routine. The original data value is not changed by this setting.

Hydraulic Status - Indicates how the node is handled when using the Trace routine.

Hydraulic Type - Specifies the hydraulic type of the associated node.

Load Units - Specifies the dimensional units for the various load values.

Name - Specifies the unique name or number for the associated node. The name may contain up to forty (40) alphanumeric characters. The name is automatically assigned when a new node is added; however, the User can change the name. Except when the *Allow Duplicate Node Names* preference settings option is selected, the name must be unique to the model. A message will be displayed if the name already exists. The value may be up to forty (40) alphanumeric characters.

Pressure - Specifies the node's pressure value. If the *Pressure* value is "known" (*Pressure Known* set to "Yes"), enter a value. If the *Pressure* value is "unknown," set the *Pressure Known* item to "No." Initially, the field is populated with the default *Node Pressure* value. If the model has been recently solved and the pressure is "unknown," the item will contain the calculated *Pressure* value for the node.

Pressure Known - Indicates whether the pressure value is "known" ("Yes") or "unknown" ("No"). If set to unknown, the *Pressure* value will be calculated during the Solution routine.

Pressure Units - Specifies the dimensional units for the *Pressure* value.

Temperature - Specifies the average flowing temperature of the gas at the node's location. If the model has been recently solved, the *Temperature* value is unknown, and the *Calculate Temperature Values* solution option is selected, the item contains the calculated *Temperature* value for the node.

Temperature Known - Indicates whether the temperature value is "known" ("Yes") or "unknown" ("No").

Gas Properties Values

[-] Gas Properties Values...	
Specific Gravity	0.6
Viscosity	0.000007
Viscosity Units	Lbm/Ft-sec
Heating Value	1000
Heating Value Units	Btu/cf
Specific Heat	1.3
Properties Known	Yes

Displayed Items

Heating Value Units - Displays the dimensional units used for the *Heating Value* value.

Viscosity Units - Displays the dimensional units used for the *Viscosity* value.

Data Items

Heating Value - Specifies the heating value of the gas at the node's location.

Properties Known - Indicates whether the gas properties are "known" ("Yes") or "unknown" ("No"). If set to unknown, the various gas property values will be calculated during the Solution routine.

Specific Gravity - Specifies the specific gravity value of the gas at the node's location. The Specific Gravity represents the weight or density of the gas compared to dry air.

Specific Heat - Specifies the specific heat ratio of the gas at the node's location. The specific heat ratio represents the constant pressure specific heat divided by the constant volume specific heat of the gas mixture.

Viscosity - Specifies the dynamic viscosity of the gas at the node's location.

GASWorkS™ 10.1

Graphic Data Items

[-] Graphic Data Items...	
X-Coordinate	-104.815565
Y-Coordinate	38.853665
Coordinate Units	Web L/L
Symbol Style	Default
Symbol Size Style	Default
Symbol Color	Default
Selected Color	
Allow Intersection Flag	Yes

Displayed Items

Coordinate Units - Displays the dimensional units used for the various coordinate values.

Data Items

Allow Intersection Flag - Indicates whether the node is included when the *Flag Unbroken Intersections* command is executed.

Selected Color - Specifies or displays the color of the node symbol. To set a color, select "Selected Color" from the *Symbol Color* list, then click in the adjacent cell. The Color Palette will be displayed. Select the desired color, then click the *OK* command button to assign the selected color.

Symbol Color - Specifies whether the "Default" color or a "Selected Color" will be used to display the node's symbol.

Symbol Size - If the *Symbol Size Style* is not set to "Default," specifies the size of the node symbol.

Symbol Size Style - Specifies the style for the Node symbol size. An "Absolute" style will cause the symbol to change in size as the model image zoom level changes. A "Relative" style will cause the symbol to be displayed at a consistent size as the zoom level changes.

Symbol Style - Specifies the style of the symbol used to display the node's location.

X-Coordinate - Specifies the node's east-west location.

Y-Coordinate - Specifies the node's north-south location.

Z-Coordinate - Specifies the node's vertical location. This item is only displayed when 3D coordinates are in use.

Text Display Items

[-] Text Display Items...	
Display Text Items	Yes
Text Location Style	User Selected
Text X, Web L/L	-104.815714
Text Y, Web L/L	38.853581
Text Rotation	Default

Displayed Items

Text Location Style - Displays the location style of the node text items.

Data Items

Display Text Items - Indicates whether the data text items for the node will be displayed. The node text items will only be displayed if the *Display Node Items* option is selected in the Text Display Settings.

Text Rotation - If the *Text Location Style* is "User Selected," specifies the rotation angle of the node text items.

Text X - If the *Text Location Style* is "User Selected," specifies the east-west location of the node text items.

Text Y - If the *Text Location Style* is "User Selected," specifies the north-south location of the node text items.

Calibration Data Values

[-] Calibration Data Values...	
Calibration Type	None

Displayed Items

Calibration Diff - Displays the difference between the known value and the value in the model.

Calibration Diff, % - Displays the difference, as a percentage, between the known value and the value in the model.

Data Items

Calibration Type - Specifies the type of calibration style.

Calibration Value - Specifies the known value to be checked against. The value is expressed in the *Pressure Units* or *Load Units*, depending on the selected *Calibration Type*.

Miscellaneous Data Values

[-] Miscellaneous Data Values...	
Pipe Count	3
Edit Date	12/24/2019

Displayed Items

Edit Date - Displays the date when the node data was last changed.

Pipe Count - Displays the number of pipes connected to the node.

Calculated Values

[-] Calculated Values...	
Pressure (Gauge)	14.90 Psi
Atmospheric Pressure	11.75 Psi
Pressure (Absolute)	26.66 Psi
Base Load	0.00 Mcfh
External Load	-1.23 Mcfh
External Load (Fixed)	0.00 Mcfh
Total Load	-1.23 Mcfh
Flow Error	0.06 Mcfh
Total Energy Load	-1232 MBtu/hr

Displayed Items

Atmospheric Pressure - Displays the calculated *Atmospheric Pressure* value based on the node's elevation. The units for this item are set the same as the *Pressure Units*.

Base Load - If the *Base Load* value is "known," displays the *Base Load* value times the Design Factor if the *Adjust Base Load* item is "Yes." If the *Base Load* value is "unknown," displays the value calculated by the Solution routine.

External Load - Displays the external load value times the Design Factor if the *Adjust External Load* item is "Yes."

External Load (Fixed) - Displays the portion of the external load that cannot be adjusted by the Design Factor.

Flow Error - Displays the difference between the total flow into the node and the total flow out of the node.

Pressure (Absolute) - Displays the calculated absolute pressure value.

Pressure (Gauge) - Displays the calculated (or entered) gauge pressure value.

Total Energy Load - Displays the total (volumetric) load value times the heating value.

Total Load - Displays the total of the (adjusted) *Base*, *External Load*, and *External Load (Fixed)* values.

Actions

Displays the various GDI Commands that may be executed from the Node Data Panel.

```
[-] Actions - Select One...
    Add Associated User Text
    Add To Favorite Results
    Delete Selected Node
    Edit Multiple Nodes
    Flag Selected Node
    Match All Node Properties
    Match Node Graphic Properties
    Match Node Hydraulic Properties
    Move Selected Node
    Move Text For Selected Node
    Print Data List Values
```

Action	*GDI Command* - See Description
Add Associated User Text	*Add User Text - Associated*
Add To Favorite Results	None - Adds selected the node to the *Favorite Results* list.
Clear Flag For Selected Node	*Clear Node Flag*
Delete Selected Node	*Delete Node*
Edit Multiple Nodes	*Edit Multiple Nodes*
Flag Selected Node	*Flag Node*
Match All Node Properties	*Match Node Properties (All)*
Match Node Graphic Properties	*Match Node Graphic Properties*
Match Node Hydraulic Properties	*Match Node Hydraulic Properties*
Move Selected Node	*Move Node*
Move Text For Selected Node	*Move Node Text*
Print Data List Values	None - Prints the contents of the data list.

Command Buttons

Apply Data Values - Saves and applies changes.

Continue - Proceed with the associated routine without making changes.

Dismiss - Closes the Data Panel when no changes have been made.

Notes & Considerations

● Data cells with a "yellow" background color display assigned or calculated values. Values in these cells cannot be directly changed by the User.

● The load for a node is comprised of two separate values: the base load and the external load. The base load represents a load value that may be directly input by the User or calculated during the Solution routine. The external load represents the load associated with the customers attached to the pipe. The external load cannot be changed directly by the User. The *External Load* value has two portions: a portion that may be adjusted by the Design Factor and a portion that cannot be adjusted by the Design Factor.

● See the *Working With An Attribute Database* topic in the *Miscellaneous* section for more information on using an attribute database.

● The Hydraulic Status is only used during the tracing functions; it does not affect the solution results.

● When a new node feature is entered, its data values are populated by one of two methods. If the *Use Last Record For Default Values* preference settings option, the data from the last node record in the model is used as the "default" data for the new feature (record). If it is the first node feature entered in the model, the values from the Default Data Settings are used as the "default" data for the new record. If the *Use Last Record For Default Values* preference settings option is not selected, the values from the Default Data Settings are used as the "default" data for all new node records.

● The "Z" coordinate value and the elevation value are not required to be the same. If 3D coordinates are used, the default elevation value will be set to the "Z" coordinate value when a new feature is entered; however, the value may be manually entered (overridden) as appropriate.

Pipe Data & Results

The data associated with the pipes may be accessed through one of the various *Edit Pipe Data* methods. The Pipe Data is displayed in the Data Panel. The features associated with the pipe data are described in the following sections.

Pipe Data & Results...	
Record Number	2
Internal ID Number	2
From Node	10
To Node	15

Displayed Items

Internal ID Number - Displays the pipe's unique (to GASWorkS) internal identification number. This value is automatically assigned and cannot be changed by the User.

From Node - Displays the name of the pipe's beginning node.

Record Number - Displays the position of the current pipe's record in the Pipe data file. This value is automatically assigned and cannot be changed by the User.

To Node - Displays the name of the pipe's ending node.

Hydraulic Data Items

The contents of the *Hydraulic Data Items* will vary depending on the selected *Hydraulic Type*. The various data items are described as follows.

[-] Hydraulic Data Items...	
Hydraulic Type	Pipe
Size/Type Code	2P
Diameter Units	Inches
Roughness	0.00006
Hydraulic Length	359.064
Length Units	Feet
Hydraulic Efficiency	0.95
Efficiency Units	Decimal
Status	On
Flow Equation	IGT-Improved
Flow Rate Units	Mcfh
Pressure Drop Units	Psi

Displayed Items

Efficiency Units - Displays the dimensional units used for the *Efficiency* values. This item is only displayed for *Compressor*, *Fitting*, and *Pipe* Hydraulic Types.

Data Items

Control Style - Specifies whether the set pressure for a compressor-type feature is to be applied to the downstream (discharge) node or the upstream (suction) node. This item is only displayed for *Compressor* Hydraulic Types.

Diameter Units - Specifies the dimensional units for the Size/Type Code value. This item is only displayed for *Pipe* Hydraulic Types.

Flow Equation - Specifies the "pipe" flow equation. This item is only displayed for *Fitting* and *Pipe* Hydraulic Types.

Flow Rate Units - Specifies the dimensional units to be used for calculated flow rate values.

Hydraulic Efficiency - Specifies the efficiency value. Initially, the efficiency is set to the default *Pipe Efficiency* value. For pipe and fitting-type features, this item refers to the Hydraulic Efficiency of the feature. For compressor-type features, this item refers to the Mechanical Efficiency of the compressor. This item is only enabled for *Compressor*, *Fitting*, and *Pipe* Hydraulic Types.

Hydraulic Length - Specifies the hydraulic length for a pipe-type feature. For graphically entered pipe features, the initial value is set to the pipe's "graphic" length. This item is only displayed for *Pipe* Hydraulic Types.

Hydraulic Type - Specifies the hydraulic element type for a pipe-type feature. This value is initially set to "Pipe."

Length Units - Specifies the length dimensional units for the associated pipe-type feature. This item is only displayed for *Pipe* Hydraulic Types.

Number Of Wells - Specifies the number of wells for the associated well-type feature. This item is only displayed for *Well* Hydraulic Types.

Pressure Drop Units - Specifies the dimensional units to use for the calculated pressure drop values.

Roughness - If the Size/Type Code is manually entered, specifies the inside pipe wall roughness value for a pipe-type feature. If the Size/Type Code is selected, displays the inside pipe wall roughness value for a pipe-type feature, and the value is extracted from the Pipe Property Table for the assigned Size/Type Code. This value is only displayed for *Pipe* Hydraulic Types.

Set Pressure - Specifies the set pressure value for a compressor or regulator-type feature. This item is only displayed for *Compressor* and *Regulator* Hydraulic Types.

Set Pressure Units - Specifies the dimensional units for the *Set Pressure* value. This item is only displayed for *Compressor* and *Regulator* Hydraulic Types.

Size/Type Code - Specifies the Size/Type Code. The meaning of this item will vary depending on the selected *Hydraulic Type*. The various meanings are listed in the following table:

Hydraulic Type	*Size/Type Code Meaning*
Compressor	Represents a code for a compressor size and type. The content of the list represents the Size/Type Codes found in the currently assigned Compressor Property Table.
Fitting	Represents a code for a fitting size and type. The content of the list represents the Size/Type Codes found in the currently assigned Fitting Property Table.

Hydraulic Type	Size/Type Code Meaning
Pipe	Represents the Size/Type Code or the *Inside Diameter* value for a pipe feature. Select the desired Size/Type Code, or enter the *Inside Diameter* value. The content of the list represents the Size/Type Codes found in the currently assigned Pipe Property Table.
Regulator	Represents a code for a regulator size and type. The content of the list represents the Size/Type Codes found in the currently assigned Regulator Property Table.
Valve	Represents a code for a valve size and type. The content of the list represents the Size/Type Codes found in the currently assigned Valve Property Table.
Well	Represents a code for a well size and type. The content of the list represents the Size/Type Codes found in the currently assigned Well Property Table. If the "Unique" item is selected, the well parameters must be manually entered in the appropriate cells.

Status - Indicates whether the *Hydraulic Type* is turned "on" or "off."

Valve Opening - Specifies the opening position for a valve-type feature. The value is expressed as a percentage, with 0 (zero) being completely closed and 100 being completely open. This item is only displayed for *Valve* Hydraulic Types.

Well Coefficient - Specifies the equation coefficient for a well-type feature. This item is only displayed for *Well* Hydraulic Types when the *Size/Type Code* is "Unique."

Well Exponent - Specifies the equation exponent for a well-type feature. This item is only displayed for *Well* Hydraulic Types when the *Size/Type Code* is "Unique."

Pipe Sizing Items

This section is only displayed when the *Hydraulic Type* is "Pipe."

Data Items

Allow Sizing - Indicates whether the pipe size will be calculated during the Solution routine when the routine is started using the *Solve & Calculate Pipe Sizes* command button.

Sizing Group - Specifies the Sizing Group to associate the current pipe-type feature with. When using the Pipe Sizing routine, only pipe sizes associated with the assigned group will be considered during the pipe selection.

Attached Fittings

This section is only displayed when the *Hydraulic Type* is "Pipe."

[-] Attached Fittings...	
Attached Fittings	0
Equivalent Length	0 Feet
Fitting	Click To Add...

Displayed Items

Attached Fittings - Displays the number of fittings attached to the pipe segment.

Equivalent Length - Displays the *Equivalent Length* value for the attached fittings. The value is calculated based on the associated values found in the current Fitting Property Table.

Data Items

Fitting - Displays an attached fitting's Size/Type code. Or, it may be used to attach a new fitting or delete or edit an existing fitting. See the following *Notes & Considerations* section for more information on its use.

Graphic Data Items

[-] Graphic Data Items...	
Pipe Line Style	Default
Pipe Line Thickness	Default
Pipe Line Color	Default
Selected Color	
Pipe Line Type	2-Point Pipe
Graphic Length	359 Feet

GASWorkS™ 10.1 Data Panel

Table Of Contents

Displayed Items

Center Point X - Displays the east-west location of the "center point" of an arc-type pipe. This item is only displayed for "Arc" graphic style pipe features.

Center Point Y - Displays the north-south location of the "center point" of an arc-type pipe. This item is only displayed for "Arc" graphic style pipe features.

Curve Radius - Displays the radius of curvature of an arc-type pipe. This item is only displayed for "Arc" graphic style pipe features.

Delta Angle - Displays the included angle of an arc-type pipe. This item is only displayed for "Arc" graphic style pipe features.

From Angle - Displays the angle from the center point to the From node end of an arc-type pipe. This item is only displayed for "Arc" graphic style pipe features.

Graphic Length - Displays the graphical length of the pipe segment.

Pipe Line Type - Displays the graphical style of the pipe segment.

Rotation - Displays the direction used to input (draw) of an arc-type pipe. This item is only displayed for "Arc" graphic style pipe features.

To Angle - Displays the angle from the center point to the To node end of an arc-type pipe. Displays the included angle (delta) of an arc-type pipe. This item is only displayed for "Arc" graphic style pipe features.

Vertex Count - Displays the number of vertices associated with a polyline-type pipe feature. This item is only displayed for "Polyline" graphic style pipe features.

Data Items

Pipe Line Color - Specifies whether a "Default" color or a "Selected Color" will be used to display the line representing the pipe segment.

Pipe Line Style - Specifies the line style used to represent the pipe segment.

Pipe Line Thickness - Specifies the thickness (width) of the line used to represent the pipe segment. The value is expressed in screen pixels.

Selected Color - Specifies or displays the color of the line representing the pipe segment. To set a color, select "Selected Color" from the *Symbol Color* list, then click in the adjacent cell. The Color Palette will be displayed. Select the desired color, then click the *OK* command button to assign the selected color.

B3PE LLC   *Engineering & Software Services*
Revision - 008, Copyright 2024, All Rights Reserved.
365

Text Display Items

[-] Text Display Items...	
Display Text Items	Yes
Text Location Style	User Selected
Text X, Web L/L	-104.816249
Text Y, Web L/L	38.853794
Text Rotation, Deg	90.000

Displayed Items

Text Location Style - Displays the location style of the pipe text items.

Data Items

Display Text Items - Indicates whether the data text items for the pipe will be displayed. The pipe text items will only be displayed if the *Display Pipe Items* option is selected in the Text Display Settings.

Text Rotation - If the *Text Location Style* is "User Selected," specifies the rotation angle of the pipe text items.

Text X - If the *Text Location Style* is "User Selected," specifies the east-west location of the pipe text items.

Text Y - If the *Text Location Style* is "User Selected," specifies the north-south location of the pipe text items.

Miscellaneous Data Values

[-] Miscellaneous Data Values...	
Facility Type	UnSpecified
MAOP, Psi	0.00
Customer Count	5
Edit Date	12/24/2019

Displayed Items

Customer Count - Displays the number of customers attached to the associated pipe feature. This item is calculated; however, the calculated value may be overridden by the User. To change the pipe's Customer Count, use the *Edit Pipe Customer Count* command. *Note -* Changing this item can corrupt the external load accounting. Particular care should be taken when changing this value.

Edit Date - Displays the date when the pipe data was last changed.

Part Of Group ID - Displays the internal identification number of the group to which the pipe feature belongs to. This item is only displayed when the pipe feature is included in a Group.

Data Items

Facility Type - Specifies the Facility Type to be associated with the pipe segment.

Group Type - Specifies the Group Type that the pipe feature belongs to. This item is only displayed when the pipe feature is included in a Group.

MAOP - Specifies the Maximum Allowable Operating Pressure associated with the pipe segment.

Attribute Data Values

If a pipe attribute file (database) is attached to the model, the *Attribute Data Values* section will display the values in the database. The values in this section will vary depending on the content of the database. If a "linked" style database is attached, the values may be changed by the User. If a "non-linked" style database is attached, the values are view-only and cannot be changed by the User. The persistent contents of the list are described as follows.

[-] Attribute Data Values...	
Link ID Number	1
Link To Database	Yes

GASWorkS™ 10.1   *Data Panel*

Displayed Items

Link ID Number - Displays the identification number used to link the pipe feature in the model to the attribute file or as an identifier for the pipe. When a "linked" style database is attached, the number is automatically assigned and cannot be changed by the User. When a "non-linked" style database is attached, the number is assigned by the User. Generally, the number should represent a unique value in the attribute file (database). When no database is attached, the number may be used to maintain any User desired alphanumeric value. The value may be a combination of up to twenty (20) alphanumeric characters.

Data Items

Link To Database - Indicates whether the pipe feature should be linked to the attribute file (database). If "Yes" is selected, the pipe will be linked to the database by matching the Link ID Number.

Calculated Values

[-] Calculated Values...	
Flow Rate	2.290 Mcfh
From Pressure	14.90 Psi
To Pressure	14.74 Psi
Pressure Drop	0.16 Psi
Average Pressure	14.82 Psi
Average Temperature	60.00 Fahrenheit
Average Compressibility	1.000000
Inside Diameter	1.917 Inches
Roughness	0.00006 Inches
Total Efficiency	0.95 Decimal
Drop Per 100 Centimetre	0.00 Psi
Hoop Stress	77.28 Psi
SMYS	1250 Psi
Percent SMYS	6.18 %
Volume	0.013 Mcf
Velocity	18 Feet/sec
Reynolds Number	33283

Displayed Items

Average Compressibility - Displays the calculated Compressibility Factor for the pipe feature. The value is based on the average pressure and average temperature values.

Average Pressure - Displays the average pressure for the pipe feature.

Average Temperature - Displays the average temperature for the pipe feature.

Coefficient Ratio - Displays the ratio of the required (calculated) valve factor to the rated valve factor for a regulator-type feature. This item is only displayed for *Regulator* Hydraulic Types.

Compression Ratio - Displays the ratio of the absolute outlet pressure to the absolute inlet pressure for a compressor-type feature. This item is only displayed for *Compressor* Hydraulic Types.

Drop Per XXX - Displays the unit pressure drop along the pipe segment for a pipe-type feature. This value is calculated by dividing the overall pressure drop across the segment by the pipe's hydraulic length and multiplying the result by the specified *Pressure Drop Per Length*. The *Pressure Drop Per Length* value may be set on the *Other* data tab on the Report Options screen. This value is only displayed for *Pipe* Hydraulic Types.

Equivalent Diameter - Displays the equivalent diameter of a fitting-type feature. The value is extracted from the current Fitting Property Table for the assigned Size/Type Code. This item is only displayed for *Fitting* Hydraulic Types.

Equivalent Length - Displays the equivalent length of a fitting-type feature. The value is extracted from the current Fitting Property Table for the assigned Size/Type Code. This item is only displayed for *Fitting* Hydraulic Types.

Estimated Cost - Displays the estimated cost of the pipe segment. This item is only displayed if a non-zero cost value is assigned to the Size/Type Code in the associated Property Table.

Estimated Fuel Usage - Displays estimated fuel usage for the "pipe" segment. This item is only displayed for *Compressor* Hydraulic Types.

Flow Rate - Displays the calculated flow rate value for the pipe feature. A positive value indicates that the flow direction is from the From Node to the To Node, and a negative value indicates that the flow direction is from the To Node to the From Node.

From Pressure - Displays the calculated (or entered) pressure value for the pipe's From Node.

Fuel Usage Rate - Displays the fuel usage rate for a compressor-type feature. The value is extracted from the current Compressor Property Table for the assigned Size/Type Code. This item is only displayed for *Compressor* Hydraulic Types.

Hoop Stress - Displays the calculated hoop stress value for a pipe-type feature. The value is based on the larger From Node or To Node pressure values and the outside diameter and wall thickness values found in the Pipe Property Table. This item is not displayed if the Size/Type Code is not in the Pipe Property Table. This value is only displayed for *Pipe* Hydraulic Types.

Inside Diameter - Displays the inside diameter value for a pipe-type feature. If the Size/Type Code represents a code, this value represents the *Inside Diameter* value extracted from the assigned Pipe Property Table. Otherwise, this item represents the interpreted numeric value of the Size/Type Code value. This value is only displayed for *Pipe* Hydraulic Types.

Minimum Differential - Displays the associated minimum differential value from the associated Property Table. This item is only displayed for *Regulator* Hydraulic Types.

Percent SMYS - Displays the ratio of the hoop stress value to the specified minimum yield strength (SMYS) value for a pipe-type feature. The value is based on the SMYS value found in the Pipe Property Table. The value is not displayed if the SMYS value is zero (0) or the Size/Type Code is not found in the Pipe Property Table. This value is only displayed for *Pipe* Hydraulic Types.

Pressure Drop - Displays the linear pressure drop for the pipe-type features (the From Node Pressure minus the To Node Pressure). *Note -* When elevation differences are present along a pipe segment, it is possible for the downstream pressure to be larger than the upstream pressure. A positive value indicates that the From Node pressure is larger than the To Node pressure, and a negative value indicates that the To Node pressure is greater than the From Node pressure. The report value represents the difference in the "gauge" pressure values.

Rated Coefficient - Displays the valve factor for a regulator-type feature. The value is extracted from the current Regulator Property Table for the assigned Size/Type Code. This item is only displayed for *Regulator* Hydraulic Types.

Rated Power - Displays the power rating for a compressor-type feature. The value is extracted from the current Compressor Property Table for the assigned Size/Type Code. This item is only displayed for *Compressor* Hydraulic Types.

Required Coefficient - Displays the required (calculated) valve factor for a regulator-type feature. This item is only displayed for *Regulator* Hydraulic Types.

Required Power - Displays the required (calculated) power for a compressor-type feature. This item is only displayed for *Compressor* Hydraulic Types.

Reynolds Number - Displays the Reynolds Number for a pipe-type feature. This item is only displayed for *Pipe* Hydraulic Types.

Roughness - Displays the inside pipe wall roughness value for pipe-type features. The value is extracted from the Pipe Property Table for the assigned Size/Type Code. This value is only displayed for *Pipe* Hydraulic Types.

SMYS - Displays the specified minimum yield strength (SMYS) value for a pipe-type feature. This value is only displayed for *Pipe* Hydraulic Types.

To Pressure - Displays the pressure value for the pipe's To Node.

Total Efficiency - Displays the total hydraulic efficiency value for a pipe-type feature. This item is only displayed for *Pipe* Hydraulic Types. See the following *Notes & Considerations* section for information on how the total efficiency is calculated.

Velocity - Displays the maximum velocity of the gas flow in a pipe-type feature. The value is calculated using the lower of the segment's end pressures, the volumetric flow rate, and the inside diameter value for the pipe segment. This value is only displayed for *Pipe* Hydraulic Types.

Volume - Displays the volume of gas contained in a pipe-type feature. The value is calculated using the average pressure value, the inside diameter, and the length of the pipe segment. This value is only displayed for *Pipe* Hydraulic Types.

Actions

Displays the various GDI Commands that may be executed from the Pipe Data Panel.

```
[-] Actions - Select One...
    Add Associated User Text
    Add To Favorite Results
    Add Vertex To Selected Pipe
    Copy Selected Pipe
    Delete Selected Pipe
    Edit Multiple Pipes
    Flag Selected Pipe
    Insert Compressor Into Selected Pipe
    Insert Regulator Into Selected Pipe
    Insert Valve Into Selected Pipe
    Join A Pipe To The Selected Pipe
    Match All Pipe Properties
    Match Pipe Facility Type
    Match Pipe Graphic Properties
    Match Pipe Hydraulic Properties
    Move An End Of The Selected Pipe
    Move Text For Selected Pipe
    Print Data List Values
    Segment Selected Pipe
```

Action	*GDI Command* **- See Description**	*Hydraulic Type*
Add Associated User Text	*Add User Text - Associated*	All
Add To Favorite Results	None - Adds selected pipe to the *Favorite Results* list.	All
Add Vertex To Selected Pipe	*Add Polyline Pipe Vertex*	2-Point Pipe Polyline Pipe
Clear Flag For Selected Pipe	*Clear Pipe Flag*	All
Copy Selected Pipe	*Copy Pipe*	All
Delete Selected Pipe	*Delete Pipe*	All

Action	*GDI Command* - **See Description**	*Hydraulic Type*
Delete Vertex From Selected Pipe	*Delete Polyline Pipe Vertex*	Polyline Pipe
Edit Multiple Pipes	*Edit Multiple Pipes*	All
Flag Selected Pipe	*Flag Pipe*	All
Insert Compressor Into Selected Pipe	*Insert Compressor*	Pipe
Insert Regulator Into Selected Pipe	*Insert Regulator*	Pipe
Insert Valve Into Selected Pipe	*Insert Valve*	Pipe
Join A Pipe To The Selected Pipe	*Join Adjacent Pipes*	All
Match All Pipe Properties	*Match Pipe Properties (All)*	All
Match Pipe Facility Type	*Match Pipe Facility*	All
Match Pipe Graphic Properties	*Match Pipe Graphic Properties*	All
Match Pipe Hydraulic Properties	*Match Pipe Hydraulic Properties*	All
Move An End Of The Selected Pipe	*Move Pipe End*	All
Move Text For Selected Pipe	*Move Pipe Text*	All
Move Vertex For Selected Pipe	*Move Polyline Pipe Vertex*	Polyline Pipe
Print Data List Values	None - Prints the contents of the data list.	All
Remove Attached Fittings	*Remove Fittings*	Pipe
Segment Selected Pipe	*Segment Pipe*	2-Point Pipe
Straighten Selected Polyline Pipe	*Straighten Polyline Pipe*	Polyline Pipe
Swap Ends Of Selected Pipe	*Swap Pipe Ends*	All
Tap Selected Pipe	*Tap Pipe*	Pipe

Command Buttons

Apply Data Values - Saves and applies changes.

**Continue -** Proceed with the associated routine without making changes.

Dismiss - Closes the Data Panel when no changes have been made.

Notes & Considerations

● Data cells with a "yellow" background color display assigned or calculated values. Values in these cells cannot be directly changed by the User.

● For pipe-type features, the *Size/Type Code* item can represent one of two values. It can represent a value that matches a Size/Type Code found in the current Pipe Property Table. Or, it can represent the actual *Inside Diameter* value of the pipe. If the value represents a matching Size/Type Code, the *Inside Diameter* value is extracted from the current Pipe Property Table when needed. If a matching Size/Type Code is not found in the Pipe Property Table, it is converted to a numeric value when the inside diameter is required.

● During the Solution routine, a "total" efficiency is computed for pipe-type features and used in the hydraulic calculation. The *Total Efficiency* value is computed as the product of the efficiency assigned directly to the pipe feature and the *Efficiency* value found in the current Pipe Property Table for the matching Size/Type Code. If no matching record is found in the current Pipe Property Table, only the pipe's assigned value is used.

● Fittings may be attached to a pipe segment. The total equivalent length of the attached fittings is added to the hydraulic length of the pipe segment during the Solution routine. The *Equivalent Length* value is based on the inside diameter and equation values. In most cases, attachment of fittings has a negligible effect on the hydraulic calculation results; however, in cases where many fittings are encountered in a short distance (like meter sets, regulator sets, or compressor yard piping), it is best to include the effect of the fittings in the hydraulic calculations, by attaching the appropriate fittings to the associated pipe feature.

● If an attribute file (database) is attached, the Link ID Number matches the selected data feature and the corresponding record in the database. The first record in the database with a matching Link ID Number is displayed in the *Attribute Data Values* section. See the *Working With An Attribute Database* topic in the *Miscellaneous* section for more information on using an attribute database.

● When a new pipe feature is entered, its data values are populated by one of two methods. If the *Use Last Record For Default Values* preference settings option is selected, the data from the last pipe record in the model is used as the "default" data for the new feature (record). If it is the first pipe feature entered in the model, the values from the Default Data Settings are used as the "default" data for the new record. If the *Use Last Record For Default Values* preference settings option is not selected, the values from the Default Data Settings are used as the "default" data for all new pipe records.

● When the *Attached Fittings* section is available:

   • To attach a new fitting to the pipe feature, click in any cell where the "Select To Add..." text is displayed.

• To delete an attached fitting, click in the cell displaying the Size/Type code for the fitting to be removed, and select "Delete Fitting..." from the resulting dropdown list.

• To change or edit an attached fitting, click in the cell displaying the Size/Type code for the fitting to be changed and select "Edit Fitting..." from the resulting dropdown list. Make the desired change. Click the *Attach* command button to save the change.

● The Group Type data item is only used for identification purposes; it does not affect on the hydraulic model or calculations. It can be used during query routines to find a specific group of features.

Settings & Actions

The Settings & Actions list may be accessed by selecting the *Settings & Actions* item from the Data Panel. The features associated with the Settings & Actions list are described in the following sections.

Settings

Displays the various routines that may be executed from the *Settings* section.

Background Image Settings - Displays the Background Image Settings screen.

Color Display Settings - Displays the Color Display Settings screen.

Facility Settings - Displays the Facility Settings screen.

Graphic Settings - Displays the Graphic Settings screen.

Set Data Panel Width - Allows the width of the Data Panel to be set by the User. Enter a value on the GDI Prompt Line. The value is expressed as a percentage of the computer screen's width.

Text Display Settings - Displays the Text Display Settings screen.

GASWorkS™ 10.1 *Data Panel*

Actions

Displays the various routines that may be executed from the *Actions* section.

Action	*GDI Command* - **See Description**
Add An Action	None - Allows an action (command) to be added to the *Actions* list.
Cancel Current GDI Command	*Cancel Current GDI Command*
Undo Last Change	*Undo Last Change*

Notes & Considerations

● The Settings & Actions list provides a quick way to access various settings and commands. A command may be added to the "Actions" portion of the list by selecting the "Add An Action" item. A list of available commands will be displayed. Select a command from the list. The selected command will be added to the Actions list. To delete a command from the Actions list, right-click on the command, then press the *Enter* key. Respond to the resulting prompt accordingly.

User Graphic Image Data

If User Graphic Images are included in the model, the data associated with the features are accessed using one of the various *Edit User Graphic Image* methods. The User Graphic Image Data is displayed in the Data Panel. The features associated with the User Graphic Image Data are described as follows.
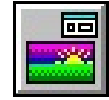
User Graphic Image Data...	
Record Number	10
Internal ID Number	3

Displayed Items

Internal ID Number - Displays the image's unique (to GASWorkS) internal identification number. This value is automatically assigned and cannot be changed by the User.

Record Number - Displays the position of the current record in the User Graphics data file. This value is automatically assigned and cannot be changed by the User.

Data Items

[-] Data Items...	
Image Name	Sample Logo.bmp
Image Width, Feet	104
Overlap Style	Combine Images
X-Coordinate	622.895
Y-Coordinate	1333.724
Coordinate Units	Feet

Displayed Items

Coordinate Units - Displays the dimensional units used for the coordinate values.

Image Name - Displays the filename of the User Graphic Image.

Data Items

Image Width - Specifies the width value of the User Graphic Image.

Overlap Style - Specifies the method used to combine or overlap images.

X-Coordinate - Specifies the east-west location of the User Graphic Image.

Y-Coordinate - Specifies the north-south location of the User Graphic Image.

Miscellaneous Data Values

[-] Miscellaneous Data Values...	
Facility Type	UnSpecified

Displayed Items

Part Of Group ID - Displays the Group ID the User Graphic Image belongs to. This item is only displayed when the User Graphic Image is included in a Group.

Data Items

Facility Type - Specifies the Facility Type to associate with the User Graphic Image.

Group Type - Specifies the Group Type the User Graphic Image belongs to. This item is only displayed when the User Graphic Image is included in a Group.

Actions

Displays the various GDI Commands that may be executed from the User Graphic Image Data Panel.

[-] Actions - Select One...
Copy Selected User Graphic Image
Delete Selected User Graphic Image
Move Selected User Graphic Image
Print Data List Values

Action	GDI Command - See Description
Copy Selected User Graphic Image	*Copy User Graphic Image*
Delete Selected User Graphic Image	*Delete User Graphic Image*
Move Selected User Graphic Image	*Move User Graphic Image*
Print Data List Values	None - Prints the contents of the data list.

Command Buttons

Apply Data Values - Saves and applies changes.

Continue - Proceed with the associated routine without making changes.

Dismiss - Closes the Data Panel when no changes have been made.

Notes & Considerations

● Data cells with a "yellow" background color display assigned or calculated values. Values in these cells cannot be directly changed by the User.

● The files containing the User Graphic Images must be in the "images" directory (folder) of the currently assigned Support Files location. The Support Files location may be set using the *Paths* item from the *Preferences* submenu of the *File* menu.

● The User Graphic Image feature may include a logo or detail in the model.

User Graphic Line Data

If User Graphic Lines are included in the model, the data associated with the features are accessed using one of the various *Edit User Graphic Line* methods. The User Graphic Line Data is displayed by using the Data Panel. The features associated with the User Graphic Line Data are described as follows.
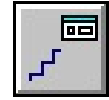
User Graphic Line Data...	▼
Record Number	9
Internal ID Number	2

Data Items

Internal ID Number - Displays the line's unique (to GASWorkS) internal identification number. This value is automatically assigned and cannot be changed by the User.

Record Number - Displays the position of the current record in the User Graphics data file. This value is automatically assigned and cannot be changed by the User.

Data Items

[-] Data Items...	
Line Style	Crossed
Pattern Spacing	10
Line Width	2
Line Color	
X1-Coordinate	1116.100
Y1-Coordinate	665.510
X2-Coordinate	964.662
Y2-Coordinate	808.946
Coordinate Units	Feet
Segment Length	209 Feet
Total Length	1575 Feet

Displayed Items

Coordinate Units - Displays the dimensional units used for the coordinate values.

Segment Length - Displays the graphic length of the selected User Graphic Line segment.

Total Length - If the line is associated with multiple segments, displays the sum of the lengths of the combined segments.

Data Items

Line Color - Specifies the color used to display the User Graphic Line. To set a color, click on the adjacent cell. The Color Palette will be displayed. Select the desired color, then click the *OK* command button to assign the selected color.

Line Style - Specifies the style used to display the User Graphic Line.

Line Width - Specifies the width used to display the User Graphic Line. The value must be greater than zero (0) and is expressed in screen pixels.

Pattern Spacing - Specifies the spacing of the symbols in certain non-continuous line styles. The value must be greater than zero (0).

X1-Coordinate - Specifies the east-west location of the start of the User Graphic Line.

X2-Coordinate - Specifies the east-west location of the end of the User Graphic Line.

Y1-Coordinate - Specifies the north-south location of the start of the User Graphic Line.

Y2-Coordinate - Specifies the north-south location of the end of the User Graphic Line.

Miscellaneous Data Values

[-] Miscellaneous Data Values...	
Facility Type	UnSpecified

Displayed Items

Part Of Group ID - Displays the Group ID the User Graphic Line belongs to. This item is only displayed when the User Graphic Line is included in a Group.

Data Items

Facility Type - Specifies the Facility Type to associate with the User Graphic Line.

Group Type - Specifies the Group Type the User Graphic Line belongs to. This item is only displayed when the User Graphic Line is included in a Group.

Actions

Displays the various GDI Commands that may be executed from the User Graphic Line Data Panel.

```
[-] Actions - Select One...
    Copy Selected User Graphic Line
    Delete Selected User Graphic Line
    Match User Graphic Line Properties
    Move Selected User Graphic Line
    Move End Of Selected User Graphic Line
    Print Data List Values
```

Action	*GDI Command* - **See Description**
Copy Selected User Graphic Line	*Copy User Graphic Line*
Delete Selected User Graphic Line	*Delete User Graphic Line*
Match User Graphic Line Properties	*Match User Graphic Line Properties*
Move Selected User Graphic Line	*Move User Graphic Line*
Move End Of Selected User Graphic Line	*Move User Graphic Line End*
Print Data List Values	None - Prints the contents of the data list.

Command Buttons

Apply Data Values - Saves and applies changes.

Continue - Proceed with the associated routine without making changes.

Dismiss - Closes the Data Panel when no changes have been made.

Notes & Considerations

● Data cells with a "yellow" background color display assigned or calculated values. Values in these cells cannot be directly changed by the User.

User Graphic Symbol Data

If User Graphic Symbols are included in the model, use one of the various *Edit User Graphic Symbol* methods. The User Graphic Symbol Data is displayed in the Data Panel. The features associated with the User Graphic Symbol Data are described as follows.

User Graphic Symbol Data...	▼
Record Number	1
Internal ID Number	1

Displayed Items

Internal ID Number - Displays the symbol's unique (to GASWorkS) internal identification number. This value is automatically assigned and cannot be changed by the User.

Record Number - Displays the position of the current record in the User Graphics data file. This value is automatically assigned and cannot be changed by the User.

Data Items

[-] Data Items...	
Symbol Style	North Arrow
Symbol Size Style	Feet
Symbol Size	100
Symbol Rotation, Deg	0.000
Symbol Color	
X-Coordinate	2308.840
Y-Coordinate	1125.490
Coordinate Units	Feet

Displayed Items

Coordinate Units - Displays the dimensional units used for the coordinate values.

Data Items

Symbol Color - Specifies the color used to display the User Graphic Symbol. To set a color, click on the adjacent cell. The Color Palette will be displayed. Select the desired color, then click the *OK* command button to assign the selected color.

Symbol Rotation - Specifies the rotation used to display the User Graphic Symbol. The value is expressed in degrees, clockwise from 12 o'clock (North). In terms of angle measurements, North is at the top of the screen. Not all User Graphic Symbols may be rotated - for those symbols, the rotation value is ignored.

Symbol Size - Specifies the size used to display the User Graphic Symbol. The size may be expressed as an absolute value in *Coordinate* units or relative to the diagonal distance across the GDI, expressed as a percentage.

Symbol Size Style - Specifies the style for the User Graphic symbol size. An "Absolute" style will cause the symbol to change in size as the model image zoom level changes. A "Relative" style will cause the symbol to be displayed at a consistent size as the zoom level changes.

Symbol Style - Specifies the style of the symbol used to display the User Graphic Symbol.

X-Coordinate - Specifies the east-west location of the center of the User Graphic Symbol.

Y-Coordinate - Specifies the north-south location of the center of the User Graphic Symbol.

Miscellaneous Data Values

[-] Miscellaneous Data Values...	
Facility Type	UnSpecified

Displayed Items

Part Of Group ID - Displays the Group ID the User Graphic Symbol belongs to. This item is only displayed when the User Graphic Symbol is included in a Group.

Data Items

Facility Type - Specifies the Facility Type to associate with the User Graphic Symbol.

Group Type - Specifies the Group Type the User Graphic Symbol belongs to. This item is only displayed when the User Graphic Symbol is included in a Group.

GASWorkS™ 10.1

Data Panel

Actions

Displays the various GDI Commands that may be executed from the User Graphic Symbol Data Panel.

```
[-] Actions - Select One...
    Copy Selected User Graphic Symbol
    Delete Selected User Graphic Symbol
    Match User Graphic Symbol Properties
    Move Selected User Graphic Symbol
    Print Data List Values
```

Action	*GDI Command* **- See Description**
Copy Selected User Graphic Symbol	*Copy User Graphic Symbol*
Delete Selected User Graphic Symbol	*Delete User Graphic Symbol*
Match User Graphic Symbol Properties	*Match User Graphic Symbol Properties*
Move Selected User Graphic Symbol	*Move User Graphic Symbol*
Print Data List Values	None - Prints the contents of the data list.

Command Buttons

Apply Data Values - Saves and applies changes.

Continue - Proceed with the associated routine without making changes.

Dismiss - Closes the Data Panel when no changes have been made.

Notes & Considerations

● Data cells with a "yellow" background color display assigned or calculated values. Values in these cells cannot be directly changed by the User.

User Text Data

If User Text features are included in the model, the data associated with the features are accessed using one of the various *Edit User Text* methods. The User Text Data is displayed in the Data Panel. The features associated with the User Text Data are described as follows.
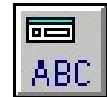
User Text Data...	
Record Number	2

Displayed Items

Record Number - Displays the position of the current record in the User Text data file. This value is automatically assigned and cannot be changed by the User.

Data Items

[-] Data Items...	
Text Value	Main St
Character Count	8
Text Rotation, Deg	90.000
Text Height	30
Text Display Style	Single Line
Text Color	
Bold	Yes
Italic	No
Underline	No
Transparent	Yes
X-Coordinate	2071.540
Y-Coordinate	1010.081
Coordinate Units	Feet

Displayed Items

Character Count - Displays the number of characters in the User Text value.

Coordinate Units - Displays the dimensional units used for the coordinate values.

Data Items

Bold - Indicates whether the User Text will be displayed using a bold (heavy) font style.

Italic - Indicates whether the User Text will be displayed using an italic font style.

Text Color - Specifies the color used to display the User Text. To set a color, click on the adjacent cell. The Color Palette will be displayed. Select the desired color, then click the *OK* command button to assign the selected color.

Text Display Style - Specifies whether the User Text will be displayed in a single line or over multiple lines.

Text Display Width - Specifies the value for the User Text width. Only displayed if "Multiline" is selected for the *Text Display Style*.

Text Height - Specifies the display height of the User Text.

Text Rotation - Specifies the rotation angle used to display the User Text. The rotation angle is measured as degrees clockwise from 12 o'clock (North). In terms of angle measurement, North is at the top of the display.

Text Value - Specifies the value to be associated with the User Text. Click in the cell to display the Edit User Text Value screen. Make the required changes, then click the *Apply* command button.

Transparent Font - When selected, the text background will be displayed transparently. When not selected, the text will be displayed with an opaque background (the model image behind the text will be "blanked out").

Underline - Indicates whether a line will be placed under the User Text.

X-Coordinate - Specifies the east-west location of the center of the User Text.

Y-Coordinate - Specifies the north-south location of the center of the User Text.

Miscellaneous Data Values

[-] Miscellaneous Data Values...	
Facility Type	**UnSpecified**

Displayed Items

Part Of Group ID - Displays the Group ID the User Text belongs to. This item is only displayed when the User Text is included in a Group.

Data Items

Facility Type - Specifies the Facility Type to associate with the User Text.

Group Type - Specifies the Group Type the User Text belongs to. This item is only displayed when the User Text feature is included in a Group.

Actions

Displays the various GDI Commands that may be executed from the User Text Data Panel.

```
[-] Actions - Select One...
    Copy Selected User Text
    Delete Selected User Text
    Match User Text Properties
    Move Selected User Text
    Print Data List Values
```

Action	GDI Command - See Description
Copy Selected User Text	*Copy User Text*
Delete Selected User Text	*Delete User Text*
Match User Text Properties	*Match User Text Properties*
Move Selected User Text	*Move User Text*
Print Data List Values	None - Prints the contents of the data list.

Command Buttons

Apply Data Values - Saves and applies changes.

Continue - Proceed with the associated routine without making changes.

Dismiss - Closes the Data Panel when no changes have been made.

Notes & Considerations

● Data cells with a "yellow" background color display assigned or calculated values. Values in these cells cannot be directly changed by the User.

Vertex Data

If features are included in the model that contains vertices (polyline pipes, branch customers, or polyline service customers), the data associated with the vertices are accessed using one of the various "edit vertex" methods. The Vertex Data is displayed in the Data Panel. The features associated with the Vertex Data are described as follows.

Vertex Data...	▼
Record Number	1

Displayed Items

Record Number - Displays the position of the current record in the Vertex data file. This value is automatically assigned and cannot be changed by the User.

Data Items

[-] Data Items...	
Vertex Style	Pipe/Main
X-Coordinate	-104.815413
Y-Coordinate	38.853666
Coordinate Units	Web L/L

Displayed Items

Coordinate Units - Displays the dimensional units used for the coordinate values.

Vertex Style - Displays the style of the vertex feature.

Data Items

X-Coordinate - Specifies the east-west location of the vertex.

Y-Coordinate - Specifies the north-south location of the vertex.

Actions

Displays the various GDI Commands that may be executed from the Vertex Data.

```
[-] Actions - Select One...
    Delete Selected Vertex
    Move Selected Vertex
    Print Data List Values
```

Action	*GDI Command* - **See Description**
Delete Selected Vertex	*Delete Polyline Pipe Vertex* *Delete Service Line Vertex*
Move Selected Vertex	*Move Polyline Pipe Vertex* *Move Service Line Vertex*
Print Data List Values	None - Prints the contents of the data list.

Command Buttons

Apply Data Values - Saves and applies changes.

Continue - Proceed with the associated routine without making changes.

Dismiss - Closes the Data Panel when no changes have been made.

Notes & Considerations

● Data cells with a "yellow" background color display assigned or calculated values. Values in these cells cannot be directly changed by the User.

● Use the *Edit Polyline Pipe Vertex* command to display the data for vertices associated with a polyline pipe. Or, if the vertex symbols are displayed, select the vertex symbol to display the associated data.

● Use the *Edit Service Line Vertex* command to display the data for vertices associated with a polyline pipe. Or, if the vertex symbols are displayed, select the vertex symbol to display the associated data.

● The vertex symbol display can be turned "on" and "off" using the *Display Vertex Symbols* graphic settings option.

● If a vertex associated with a polyline pipe is moved, respond to the "update length" prompt accordingly. Use the *Edit Service Line Vertex* command to display the data for vertices associated with a polyline pipe. Or, if the vertex symbols are displayed, select the vertex symbol to display the associated data.

● When a pipe or customer (service line) is selected that contains vertices, and if the grip symbols are displayed, the vertex location may be moved by clicking on the grip symbol, moving the symbol to the new location, and clicking at the new location. The grip symbol display and size can be controlled using settings and options in the Graphic Settings.

PROPERTY TABLES

GASWorkS™ 10.1   *Property Tables*

Table Of Contents

Using The Property Tables

The Property Table Report feature allows redundant pipe data to be included in a common file and helps support the modeling of non-pipe hydraulic features. The Property Tables use relational data management techniques to maintain a relation between the pipe data files and the Property Tables. A Property Table may be thought of as a reference table. Information may be stored in the Property Table and then referenced to or "looked up" based on a common link value (Size/Type Code). The Size/Type Code establishes the link between the pipe data files and the Property Tables. The pipe data and the data contained in the Property Tables are said to be "related" by the Size/Type Code. Through this relation, GASWorkS can access the data found in the Property Tables as needed without storing all of the model-required data in the pipe data files.

Several items in the pipe data file may be linked to a single item in the Property Table. This feature allows redundant data to be stored only once, reducing data storage and entry requirements. This feature also allows mass changes to be made by changing the value one time in the Property Table and automatically changing all of the pipe-type features that are related to that particular Property Table and item.

To demonstrate how Property Tables may be used, consider the Pipe Property Table. This Property Table contains information about the dimensional properties of various pipe sizes and types. For each unique Size/Type Code, the Property Table stores the wall thickness, outside diameter, and calculated inside diameter of the pipe. The diameter for a pipe record may be entered as a Size/Type Code value instead of entering the actual inside pipe diameter. For example, the code "2P" (which represents two-inch plastic) may be input for a pipe's Size/Type (diameter) instead of an actual *Inside Diameter* value of "1.917." When an actual inside diameter for the pipe is required, the Pipe Property Table will be searched for the Size/Type Code value equal to "2P." When a match is found, the value of the inside diameter associated with that record is extracted from the Pipe Property Table.

There are several advantages to using the Property Tables to store the dimensional pipe data. First, it eliminates the need for the User to remember the specific values for the various data items in their system. By defining the Property Table using logical codes, like "2P," to represent a two-inch plastic pipe, the User only needs to remember these easily identified Size/Type Codes. Second, the use of Property Tables helps to ensure the integrity of the model data. By maintaining the pipe sizes in a single location and selecting the Size/Type Codes from the list when entering data, entry errors may be reduced, and consistent size values may be assured.

"Standard" Property Tables are included when GASWorkS is installed. The content of the various Property Tables may be modified as necessary to meet the User's specific needs. The Property Tables may be revised, added to, or deleted by the User. When choosing values to be used as the Size/Type Codes for the data items, attempt to choose codes that are short and logical. If you choose codes that are long or hard to understand, you have defeated one of the biggest advantages of using the Property Table features.

To view or edit the contents of the Property Table Report, select the *Property Tables* item from the *Edit* menu. The Property Table Report will be displayed. Select the data tab associated with the desired Property Table.

Updating The Property Tables

The "standard" Property Tables provided when GASWorkS is installed provide a basis for building your Property Tables. The values in the "standard" Property Tables may be modified, new items may be added, and non-applicable items may be deleted. The data values associated with each Property Table may be viewed or printed by selecting the *Property Tables* item from the *Edit* menu.

Note that items are continually added to the "standard" Property Tables as Users request additional items. New and revised Property Tables are posted on our website, www.b3pellc.com, and may be downloaded to replace or merge with the existing Property Tables.

Some Property Tables contain specific data for a particular group of related items. For example, one Property Table contains only size information for copper tubing. The User may select only the Property Tables and items appropriate for their type of system or work. Multiple Property Tables may be combined to create a single comprehensive Property Table using the Merge routine in the Property Tables.

GASWorkS™ 10.1   *Property Tables*

Property Table Editor

The data in the various Property Tables may be viewed, printed, and edited using the Property Table Editor. The content displayed in the table will change depending on the selected type. The general features associated with the Property Table Editor and the various data tabs (Property Tables) are described in the following sections.
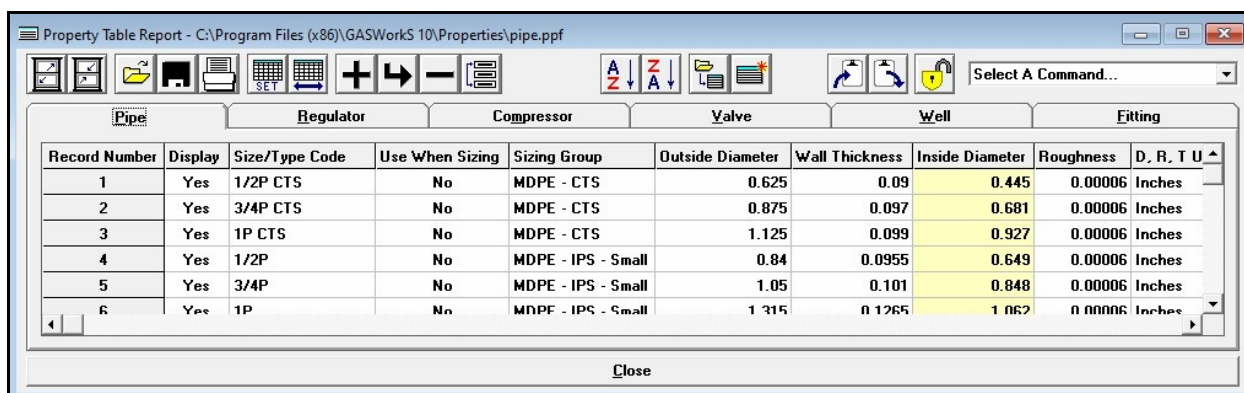
Displayed Items

Record Number - Displays the Record Number of the associated Size/Type Code displayed in the Property Table. The values are used for reference only and cannot be changed by the User.

Select A Command - Used to select various commands. The available commands are:

● The *Set All Display To 'Yes'* item will change the *Display* value for all records to "Yes."

● The *Set All Display To 'No'* item will change the *Display* value for all records to "No."

● The *Set All Use When Sizing To 'Yes'* item will change the *Use When Sizing* value for all records to "Yes." This command is only displayed on the Pipe Property Table.

● The *Set All Use When Sizing To 'No'* item will change the *Use When Sizing* value for all records to "No." This command is only displayed on the Pipe Property Table.

● The *Set All Sizing Group To 'None'* item will change the *Sizing Group* value for all records to "None." This command is only displayed on the Pipe Property Table.

● The *Set Sizing Group List* item will change the *Sizing Group* value for all records to a User-selected item. This command is only displayed on the Pipe Property Table.

Icons

Add Record - Adds a new record to the end of the current Property Table.

Copy To Clipboard - Copies the contents of the current Property Table to the Windows Clipboard. Once copied, the contents may be pasted into another application.
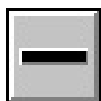
Delete Record - Deletes the selected (highlighted) record in the current Property Table.

Insert From Clipboard - Inserts data from the Windows Clipboard. The data will be inserted into the current Property Table immediately after the selected (highlighted) record. The data must be in a tab-delimited format that only contains "data" values and does not include header titles or record numbers. Generally, it is best to insert data that has been previously copied from a Property Table using the *Copy To Clipboard* icon.
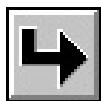
Insert Record - Inserts a new record just before the selected (highlighted) record in the current Property Table.

Locked Table - Indicates that the Property Table is locked from editing. Click this icon to "unlock" the table to allow editing. Once clicked, the contents of a previously locked Property Table may be changed.
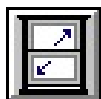
Maximize Screen Size - Increases the Property Table's size to fill the GASWorkS display.

Merge Table - Merges the contents of another Property Table into the current Property Table. When clicked, a message will be displayed asking whether to insert the new data at the selected (highlighted) record location (click the *Yes* command button) or at the end of the Property Table (click the *No* command button) - respond appropriately. The File Selection screen is displayed. Select the name of the desired file, then click the *Continue* command button. A message will be displayed when the process is complete - click the *OK* command button to clear the message.

Move Record - Moves the selected (highlighted) record to a new location in the current Property Table. When clicked, a message will be displayed to select the new location. Click the *OK* command button to clear the message, then select the desired location.

GASWorkS™ 10.1   *Property Tables*

Open - Opens a previously saved Property Table file. When clicked, the File Selection screen is displayed.

Print - Prints the contents of the current Property Table. When clicked, the Print Settings screen is displayed.
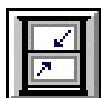
Reset Screen Size - Sets the Property Table's size to its "normal" dimensions and position.

Restore Columns - Restores the display of all columns (data items). When clicked, all of the columns associated with the current Property Table will be displayed.
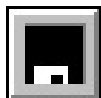
Save - Saves the contents of the current Property Table. When clicked, the File Selection screen is displayed. Once saved, the file may be opened and used with other models.

Set Columns - Used to specify which columns (data items) are displayed on the current Property Table. When clicked, the Columns specification screen is displayed.

Set As Current - Sets the current Property Table as the currently assigned Property Table in the Preference Settings. A message will be displayed when the Property Table has been set. Click the *OK* command button to clear the message.

Sort Ascending - Sorts the records in ascending order (A to Z or lowest to highest value) based on the values in the highlighted column. *Note -* If the Property Table is saved after the sort has been implemented, the Property Table will be permanently saved in the new order.

Sort Descending - Sorts the records in descending order (Z to A or highest to lowest value) based on the values in the highlighted column. *Note -* If the Property Table is saved after the sort has been implemented, the Property Table will be permanently saved in the new order.

Unlocked Table - Indicates that the Property Table is unlocked for editing. Click this icon to "lock" the table from editing. Once clicked, the table contents may be viewed; however, the contents cannot be changed until the Property Table is "unlocked."

Command Buttons

Close - Closes the report. If changes have been made to the Property Tables, a message will be displayed asking whether to save the changes - respond appropriately.

Notes & Considerations

● If the content of the Property Table is too wide to print, reduce the column size, set the printer to "landscape" mode, and reduce the printer Font and Size. If the contents of the Property Table will not fit on a single page width, multiple pages will be used to print the Property Table contents.

● In a "Fittings" table, the dimensional units for the *Equivalent Diameter* and *Equivalent Length* values are independent of the diameter and length units used for the pipes. In other words, the Property Table dimensional units may differ from the units used for the pipes. For example, the *Equivalent Diameter* values might be expressed in "Inches," and the pipe *Diameter* values might be expressed in "Centimetres." The appropriate unit conversion will automatically take place when using and reporting the Property Table values.

● In a "Pipe" table, the dimensional units for the *Outside Diameter* and *Wall Thickness* values are independent of the diameter units used for the pipes. In other words, the Property Table dimensional units may differ from the diameter units used for the pipes. For example, the Property Table values might be expressed in "Inches," and the pipe *Diameter* values might be expressed in "Millimetres." The appropriate unit conversion will automatically take place when using and reporting the Property Table values.

● During the Solution routine, a "total" efficiency is computed for *Pipe* and *Fitting* Hydraulic Types. This value is used in the hydraulic calculations. The *Total Efficiency* value is computed as the product of the efficiency assigned directly to the pipe segment, and the *Efficiency* value in the Pipe Property Table for the matching Size/Type Code. If no matching record is found in the Pipe Property Table, only the assigned value is used. To apply only the values in the Property Table Report, set the pipe segment value to one (1). To apply only the pipe segment value set the associated Property Table value to one (1).

GASWorkS™ 10.1 *Property Tables*

Compressor Tab

Data Items

Cost Per Power Unit - Specifies the cost per power unit for the associated compressor. Any currency unit may be used. The value is expressed in the *Power Units*.

Description - Specifies a brief description of the associated compressor. The description may contain up to eighty (80) alphanumeric characters.

Display - Indicates whether the associated compressor record will be included in the Size/Type Code selection list. Only items where the value of the *Display* item is "Yes" will be displayed in the lists.

Equation - Specifies the power equation that will be used to describe the compressor's power-pressure-flow characteristics. See the *Compressor Power Equations* topic in the *Appendix* section for a complete description of the equations.

Fuel Consumption - Specifies the fuel consumption rate for the associated compressor. The value is expressed in the *Fuel Units*.

Fuel Units - Specifies the dimensional unit for the fuel consumption rate value.

Parameter 1 - The definition of values in this column varies depending on the selected power equation. The following table summarizes the definition of this item as associated with each of the power equations:

Power Equation	*Parameter 1 Definition*
Adiabatic	Not used
Empirical	K1 - Equation Constant

Power Equation	*Parameter 1 Definition*
Rule-Of-Thumb	Not used

Parameter 2 - The definition of values in this column varies depending on the selected power equation. The following table summarizes the definition of this item as associated with each of the power equations:

Power Equation	*Parameter 2 Definition*
Adiabatic	Not used
Empirical	K2 - Equation Constant
Rule-Of-Thumb	Not used

Parameter 3 - The definition of values in this column varies depending on the selected power equation. The following table summarizes the definition of this item as associated with each of the power equations:

Power Equation	*Parameter 1 Definition*
Adiabatic	Not used
Empirical	K3- Equation Constant
Rule-Of-Thumb	Not used

Part Number - Specifies the associated part number for the associated compressor.

Power Units - Specifies the dimensional units for the rated power value.

Rated Power - Specifies the rated power of the associated compressor. The rated power is only used as a comparison value by the Report and Display routines.

Reference - Specifies the reference source for the information entered for the associated compressor. The reference may contain up to eighty (80) alphanumeric characters.

Size/Type Code - Specifies the unique Size/Type Code for the associated compressor. The Size/Type Code may be any group of alphabetic, numeric, or special characters up to twenty (20) characters in length. Use a logical code to identify the value. This item relates the Compressor Property Table to the pipe data files.

Specification - Specifies the specification document (PDF) for the information entered for the associated compressor. The specification may contain up to eighty (80) alphanumeric characters.

Notes & Considerations

● The values of the compressor parameters must be based on an equation whose dimensional units are consistent with the following units:

Equation Parameter	Dimensional Units
Pressure or Pressure Drop	Psia
Temperature	Rankine
Volumetric Flow Rate	Cfh

● When the *Adjust Compressor Control Pressures* solution option is selected, the calculated flow and pressure values occurring at the inlet and outlet of a compressor feature are reviewed, and the control pressure is adjusted to reflect the discovered conditions.

GASWorkS™ 10.1 *Property Tables*

Fitting Tab

Data Items

Cost Per Each - Specifies the cost of the associated fitting. Any currency unit may be used.

Cv Factor - Specifies the value of the Cv Factor of the associated fitting. This item is only enabled when the selected *Pressure Drop Method* is "Crane-Cv."

Description - Specifies a brief description for the associated fitting. The description may contain up to eighty (80) alphanumeric characters.

Display - Indicates whether the associated fitting record will be included in the Size/Type Code selection list. Only items where the value of the *Display* item is "Yes" will be displayed in the lists.

D, R Units - Specifies the dimensional units for the equivalent diameter (D) and roughness ® values. This item is only enabled when the selected *Pressure Drop Method* is "Crane-K" or "Equivalent Length."

DP Units - Specifies the dimensional units for the *Pressure Drop* value. This item is only enabled when the selected *Pressure Drop Method* is "Fixed Dp."

Efficiency - Specifies the efficiency value to be used with the associated fitting. See the following *Notes & Considerations* section for additional information on using the *Efficiency* value in the Solution routine.

Equation Constant - Specifies the value of the equation constant of the associated fitting. This item is only enabled when the selected *Pressure Drop Method* is "Generic Dp2."

Equation Exponent - Specifies the value of the equation exponent of the associated fitting. This item is only enabled when the selected *Pressure Drop Method* is "Generic Dp2."

Equivalent Diameter **-** Specifies the value of the equivalent diameter for the associated fitting. The equivalent diameter represents the *Inside Diameter* value used as the basis for calculating the equivalent length. This item is only enabled when the selected *Pressure Drop Method* is "Crane-K" or "Equivalent Length." The value is expressed in the *D, R Units*.

Equivalent Length **-** Specifies the value of the calculated equivalent length of the associated fitting. This item is only enabled when the selected *Pressure Drop Method* is "Equivalent Length." The value is expressed in the *L Units*.

K Factor - Specifies the value of the K Factor of the associated fitting. This item is only enabled when the selected *Pressure Drop Method* is "Crane-K."

Length Units - Specifies the dimensional units for the *Equivalent Length* value. This item is only enabled when the selected *Pressure Drop Method* is "Equivalent Length."

Part Number - Specifies the associated part number for the associated fitting.

Pressure Drop - Specifies the value of the pressure drop of the associated fitting. This item is only enabled when the selected *Pressure Drop Method* is "Fixed Dp." The value is expressed in the *DP Units*.

Pressure Drop Method - Specifies the pressure drop method for the associated fitting.

Reference - Specifies the source for the entered information for the associated fitting. The reference may contain up to eighty (80) alphanumeric characters.

Roughness - Specifies the value of the inside wall roughness to be used for the associated fitting. This item is only displayed when the selected *Pressure Drop Method* is "Equivalent Length." The value is expressed in the *D, R Units*.

Size/Type Code - Specifies the unique Size/Type Code for the associated fitting. The Size/Type Code may be any group of alphabetic, numeric, or special characters up to twenty (20) characters in length. Use a logical code to identify the value. For example, 2 L-90 might represent a two-inch ninety-degree elbow. This item relates the Fitting Property Table to the pipe data files.

Specification - Specifies the specification document (PDF) for the entered information for the associated compressor. The specification may contain up to eighty (80) alphanumeric characters.

Symbol Style - Specifies the style of the symbol used to display the associated fitting.

Notes & Considerations

● A *Fitting* Hydraulic Type may be used to model any device for which an equivalent length may be calculated.

● The *Equivalent Length* value may be found in manufacturer tables, in a piping design handbook, or by calculation. Most equivalent lengths are expressed in terms of the length of the standard wall thickness of the pipe. It is important to know and correctly set the *Equivalent Diameter* value to calculate the *Equivalent Length* value correctly.

● Fittings are modeled as a pipe segment using the equivalent length for the pipe's length and the equivalent diameter for the pipe's diameter.

● When a *Symbol Style* is selected, the selected symbol will be displayed near the associated pipe (fitting) feature in the GDI Image.

Pipe Tab

Data Items

Cost Per Length - Specifies the unit value of the pipe's installation cost. Any currency unit may be used. The value should be consistent with the length units used with the pipe segments. For example, if the length units are "Feet," the Cost Per (Unit) Length should be expressed in cost per foot.

Description - Specifies a brief description of the associated pipe. The description may contain up to eighty (80) alphanumeric characters.

Display - Indicates whether the associated pipe record will be included in the Size/Type Code selection list. Only items where the value of the *Display* item is "Yes" will be displayed in the lists.

D, R, T Units - Specifies the dimensional units for the wall thickness (T), the outside and inside diameters (D), and the roughness ® values.

Efficiency - Specifies the efficiency value to be used with the associated pipe. See the following *Notes & Considerations* section for additional information on using efficiency in the Solution routine.

Equation - Specifies the flow equation for the associated pipe-type feature.

Outside Diameter - Specifies the value of the outside diameter of the associated pipe. The value is expressed in the *D, R, T Units*.

Part Number - Specifies the associated part number for the associated pipe segment.

Pipe Line Color - Specifies the color to display the associated pipe when the *Pipe Line Color* value in the Pipe Data of a related pipe feature is "Pipe Property Table." Either select "Default" or to set a Color, select "Click To Select Color...." The Color Palette will be displayed. Select the desired color, then click the *OK* command button to assign the selected value.

Pipe Line Style - Specifies the pipe line style to display the associated pipe, when the *Pipe Line Style* value in the Pipe Data of a related pipe feature is set to "Pipe Property Table."

Pipe Line Thickness - Specifies the pipe line thickness (width) to display the associated pipe when the *Pipe Line Thickness* value in the Pipe Data of a related pipe feature is set to "Pipe Property Table."

Reference - Specifies the source for the entered information for the associated Pipe. The reference may contain up to eighty (80) alphanumeric characters.

Roughness - Specifies the value of the inside wall roughness to be used for the associated pipe. The value is expressed in the *D, R, T Units*.

Size/Type Code - Specifies the unique Size/Type Code for the associated pipe. The Size/Type Code may be any group of alphabetic, numeric, or special characters up to twenty (20) characters in length. Use a logical code to identify the value. For example, 2P might represent a two-inch plastic pipe. This item relates the Pipe Property Table to the pipe data files.

Sizing Group - Specifies the Sizing Group for the associated pipe. Sizes may be assigned to a group - for example, plastic sizes might be assigned to one group and steel to another. Each pipe segment may be assigned to a specific group. When using the various Sizing routines, only pipe sizes associated with the assigned group will be considered during the pipe selection. The available types are described in the following *Notes & Considerations* section.

SMYS - Specifies the specified minimum yield strength value to be used with the associated pipe. The SMYS value is used for reporting purposes only. The value is expressed in the *Stress Units*.

Specification - Specifies the specification document (PDF) for the entered information for the associated pipe segment. The specification may contain up to eighty (80) alphanumeric characters.

Stress Units - Specifies the dimensional units for the SMYS (Specified Minimum Yield Strength) value.

Use When Sizing - Indicates whether the associated pipe will be available with the various Sizing routines. Only pipe types identified for use (set to "Yes") will be considered when finding the required pipe sizes for the modeled system. The available values are described in the *Notes & Considerations* section.

Wall Thickness - Specifies the wall thickness value for the associated pipe. The value is expressed in the *D, R, T Units*.

Calculated Values

Inside Diameter - Displays the *Inside Diameter* value of the associated pipe. The value is calculated using the *Wall Thickness* and *Outside Diameter* values. This value cannot directly be changed by the User. To change the value of the inside diameter, modify the *Wall Thickness* or *Outside Diameter* values. The value is expressed in the *D, R, T Units*. The calculated value may be overridden by double-clicking in the associated cell, responding to the prompt accordingly, and entering the desired value.

Notes & Considerations

● The Cost Per Length is used in the System Summary Report to calculate the "cost" of the system. Costs of various system configurations may be compared to determine the cost and benefits of various scenarios.

● The Specified Minimum Yield Strength (SMYS) value only applies to steel pipe types. However, for plastic pipe types, the Long Term Hydrostatic Strength or Hydrostatic Design Basis may be used instead of the Specified Minimum Yield Strength to estimate the Hoop Stress to Strength ratio.

● The UKDM Pipe Property Table contains pipe size/type items common to the UK gas industry. The sizes were compiled from various sources. Some items in the UKDM Pipe Property Table do not include actual *Outside Diameter* and *Wall Thickness* values. This is because the source only listed an *Inside Diameter* value. The outside diameter and wall thickness shown in the UKDM Pipe Property Table were selected to ensure that the appropriate *Inside Diameter* value would be contained in the UKDM Pipe Property Table.

● When selecting values for the diameter and wall thickness items, if manufacturing tolerances are available, consider using the smallest possible outside diameter and largest possible wall thickness, resulting in the most conservative *Inside Diameter* value.

Regulator Tab

Displayed Items

Droop Units - Displays the style and dimensional units for the droop value of the associated regulator. Currently, only the "percent" style is used.

Data Items

Cost Per Each - Specifies the cost of the associated regulator. Any currency unit may be used.

Description - Specifies a brief description of the associated regulator. The description may contain up to eighty (80) alphanumeric characters.

Differential **P**ressure **Units -** Specifies the dimensional unit for the *Differential Pressure* value.

Display - Indicates whether the associated regulator record will be included in the Size/Type Code selection list. Only items where the value of the *Display* item is set to "Yes" will be displayed in the lists.

Droop - Specifies the droop value (reduction in outlet pressure) of the associated regulator.

Equation - Specifies the flow equation that will be used to describe the regulator's pressure-flow characteristics. See the *Regulator Flow Equations* topic in the *Appendix* section for a complete description of the equations. The following table summarizes the available regulator equations and their general application by manufacturer:

Flow Equation	*Manufacturer*
American	American Meter - Non-rubber element models

Flow Equation	Manufacturer
American AFV	American - Axial Flow Valve
Equimeter	Same as Rockwell
Fisher	Fisher Controls
Grove 80	Grove - Model 80 Flexflo
Grove 900	Grove - Model 900
ISA-S75.01	American Meter - Radial Flow Valve Other regulators and control valves meeting the Instrument Society of America - Standard S75.01
Itron	Itron
Mokveld	Mokveld Valves
Mooney	Mooney Controls
Pietro Fiorentini	Pietro Fiorentini
Rockwell	Equimeter Invensys Sensus Sprague Rockwell
Table	None - Use this equation to indicate the capacity data is <u>only</u> contained in a regulator capacity table.
Universal	Peco Others - Use this equation with older style devices using the original definition for the "universal gas sizing" equation to define their pressure-flow relationship.

Minimum **Diff**erential **P**ressure **-** Specifies the minimum differential pressure (to achieve full opening) of the associated regulator. The value is expressed in the *Diff P Units*.

Parameter 1 - The definition of values in this column varies depending on the selected flow equation. The following table summarizes the definition of this item based on the various flow equations:

Flow Equation	*Parameter 1 Definition*
American	Not Used
American AFV	Xt - Critical Flow Factor
Equimeter	Not Used
Fisher	C1 - Valve Recovery Factor
Grove 80	Xt - Critical Flow Factor
Grove 900	Xt - Critical Flow Factor
ISA-S75.01	Xt - Critical Flow Factor
Itron	Not Used
Mokveld	F_L - Valve Recovery Coefficient
Mooney	C1 - Valve Recovery Factor
Pietro Fiorentini	K1 = Body Shape Factor.
Rockwell	Not Used
Table	Not Used
Universal	C1 - Valve Recovery Coefficient

Part Number - Specifies the part number for the associated regulator.

Reference - Specifies the source for the entered information for the associated regulator. The reference may contain up to eighty (80) alphanumeric characters.

Size/Type Code - Specifies the unique Size/Type Code for the associated regulator. The Size/Type Code may be any group of alphabetic, numeric, or special characters up to twenty (20) characters in length. Use a logical code to identify the value. For example, FC 1098EGR 1 could represent a Fisher Controls 1" Model 1098EGR Regulator. This item relates the Regulator Property Table to the pipe data files.

Specification - Specifies the specification document (PDF) for the entered information for the associated regulator. The specification may contain up to eighty (80) alphanumeric characters.

Valve Factor - Specifies the published "regulating" valve coefficient of the associated regulator. The following table summarizes the applicable values based on each specific flow equation:

Flow Equation	*Applicable Coefficient*
American	C - Orifice Constant
American AFV	Cv Factor
Equimeter	K Factor - Regulator Constant
Fisher	Cg (Regulating) - Gas Sizing Coefficient
Grove 80	C1 - Critical Flow Factor
Grove 900	Cv - Valve Capacity Factor
ISA-S75.01	Cv - Valve Flow Coefficient
Itron	K Factor - Regulator Constant
Mokveld	Cv - Valve Capacity Coefficient
Mooney	Cg (Regulating) - Gas Sizing Coefficient
Pietro Fiorentini	Cg - Gas Sizing Coefficient
Rockwell	K Factor - Regulator Constant
Table	Not Used
Universal	Cv - Liquid Sizing Coefficient

Notes & Considerations

● Use the *Regulator* Hydraulic Type to model regulators operating in the "control" or "working monitor" mode.

● When the *Adjust Regulator Outlet Pressures* solution option is selected, the calculated flow and pressure values occurring at the inlet and outlet of a regulator feature are reviewed, and the outlet pressure is adjusted to reflect the discovered conditions. One of the adjustments accounts for reduced outlet pressure due to droop using the droop values listed in the Regulator Property Table.

● The values of the regulator parameters must be based on a flow equation whose dimensional units are consistent with the following units:

Equation Parameter	*Dimensional Units*
Pressure or Pressure Drop	Psia
Temperature	Rankine
Volumetric Flow Rate	Cfh

GASWorkS™ 10.1 *Property Tables*

Valve Tab

Data Items

Cost Per Each - Specifies the cost per each associated valve. Any currency unit may be used.

Description - Specifies a brief description of the associated valve. The description may contain up to eighty (80) alphanumeric characters.

Diameter **Units -** Specifies the dimensional unit for the *Diameter* value.

Display - Indicates whether the associated valve record will be included in the Size/Type Code selection list. Only items where the value of the *Display* item is set to "Yes" will be displayed in the lists.

Equation - Specifies the flow equation that will be used to describe the valve's pressure-flow characteristics. See the *Valve Flow Equations* topic in the *Appendix* section for a complete description of the equations. The following table summarizes the available equations and their general application:

Manufacturer	*Applicable Flow Equation*
Generic Check Valve	Check
Generic Fittings, Valves, & Regulators	Crane
Rockwell Valve	Rockwell

Equivalent Diameter **-** Specifies the equivalent diameter of the associated valve. The value is expressed in the *D Units*. The physical definition of this item varies according to the selected flow equation. The following table summarizes the item definition associated with each of the individual flow equations:

Flow Equation	Applicable Parameter
Check	The inside diameter of the inlet pipe connection. Uses the Crane valve equation to calculate the pressure drop associated with flow through the valve.
Crane	Varies. See the Crane Publication No. 410 for a detailed description. For approximate calculations, use the inside diameter of the inlet pipe connection.
Rockwell	The inside diameter of the inlet pipe connection.

Parameter 1 - The definition of values in this column varies depending on the selected flow equation. The following table summarizes the definition of this item based on the various flow equations:

Flow Equation	Parameter 1 Definition
Check	Not Used
Crane	Not Used
Rockwell	Not Used

Part Number - Specifies the part number for the associated valve.

Reference - Specifies the source for the entered information for the associated valve. The reference may contain up to eighty (80) alphanumeric characters.

Size/Type Code - Specifies the unique Size/Type Code for the associated valve. The Size/Type Code may be any group of alphabetic, numeric, or special characters up to twenty (20) characters in length. Use a logical code to identify the value. For example, RW 2-44157 could represent a Rockwell Model 441 257S 2-inch body regulator. This item relates the Valve Property Table to the pipe data files.

Specification - Specifies the specification document (PDF) for the entered information for the associated valve. The specification may contain up to eighty (80) alphanumeric characters.

Valve Factor - Specifies the published valve coefficient for the associated valve. The definition of this item varies depending on the selected flow equation. The following table summarizes the applicable values based on each specific flow equation:

Flow Equation	Applicable Coefficient
Check	K - Resistance Coefficient
Crane	K - Resistance Coefficient
Rockwell	"F" Factor - Pressure Drop Factor

Notes & Considerations

● The *Valve* Hydraulic Type may be used to model fittings or valves where pressure-flow characteristics follow one of the available equations.

● The values of the valve parameters must be based on an equation whose units are consistent with the following units:

Equation Parameter	Dimensional Units
Pressure or Pressure Drop	Psia
Temperature	Rankine
Volumetric Flow Rate	Cfh

● When using a "check" type valve, the From node of the associated pipe feature must represent the inlet side of the valve, and the To node must represent the outlet side of the valve.

GASWorkS™ 10.1   *Property Tables*

Well Tab

Data Items

Coefficient - Specifies the value of the equation coefficient (constant) of the associated well.

Description - Specifies a brief description of the associated well. The description may contain up to eighty (80) alphanumeric characters.

Display - Indicates whether the associated well record will be included in the Size/Type Code selection list. Only items where the value of the *Display* item is set to "Yes" will be displayed in the lists.

Exponent - Specifies the value of the equation exponent of the associated well.

Reference - Specifies the source for the entered information for the associated well. The reference may contain up to eighty (80) alphanumeric characters.

Size/Type Code - Specifies the unique Size/Type Code for the associated well. The Size/Type Code may be any group of alphabetic, numeric, or special characters up to twenty (20) characters in length. Use a logical code to identify the value. This item relates the Well Property Table to the pipe data files.

Notes & Considerations

● Only one well equation is supported. It is described in the *Well Equation* topic in the *Appendix* section. The equation Coefficient and equation Exponent are well dependent. Their values are determined by performing a multipoint curve fit of data derived from a standard performance test of the well.

● The Well Property Table is useful for maintaining generic or assumed well performance data for untested or similar-performing wells. The equation Coefficient (constant) and Exponent can also be uniquely entered for an individual well-type feature without using the Property Table. To allow the entry of unique values for a well-type feature, select the "Unique" item from the *Size/Type Code* list in the Pipe Data for the well.

● The values of the well parameters must be based on an equation whose units are consistent with the following units:

Equation Parameter	*Dimensional Units*
Pressure or Pressure Drop	Psia
Temperature	Rankine
Volumetric Flow Rate	Cfh

ATTRIBUTE DATA

Attribute Data

GASWorkS allows non-model data to be attached to a model through of an external database. This non-model data is referred to as Attribute Data. Attribute Data may be attached to customer, node, and pipe features.

A customer attribute file might contain information about the customer's address, customer classification, or usage information. Usage information in a customer attribute file may be used to assign load values to the associated customer features and subsequently to the nodes in the model. A pipe attribute file might contain information about the installation date, leak history, or test data. A node attribute file might contain information about the loads.

Two styles of attribute files may be used with GASWorkS. One style is a "linked" style file, which may be defined by the User but is managed by GASWorkS. The other style is a "non-linked" style file, which is managed by a third-party application (for example, a database or spreadsheet application). When a "linked" file is attached, GASWorkS adds and deletes records from the file as features are added and deleted from the model. When a "non-linked" file is used, record maintenance must be handled by some means outside of GASWorkS.

The records in an attached attribute file are related to the associated customer and pipe model features using the *Link ID Number* item. If a "linked" style database is attached, GASWorkS handles the assignment of the Link ID Numbers. If a "non-linked" style database is attached, the User must manually assign the Link ID Numbers. Generally, the Link ID Number should be unique within the attribute file (database). The records in an attached attribute file are related to the associated node model features using the *Name* item for "non-linked" databases and the *Internal ID Number* for "linked" databases.

The Update External Loads routine causes the *External Load* item of the node data to be updated using the values found in the specified field of an attached customer attribute file. The processing behavior of the command varies depending on whether the model contains customer features. When customer features are included in the model, and the *Total Load Field* is assigned, the routine looks for a matching record in the attribute file, based on the Link ID Number, for each customer feature in the model. If a match is made, the customer load is updated with the value found in the attribute file. When all customer loads have been updated, the node external loads are updated with the new customer loads based on the *Node Load Application* method assigned to each customer.

Linked Attribute Database Definition Screen

When using a "linked" style attribute file (database), GASWorkS automatically creates and maintains a database containing attribute data as defined in the associated Attribute Database Definition. The Attribute Database Definition specifies various information about the fields to include in the attribute file. An Attribute Database Definition may be predefined, and used and shared between models. The Attribute Database Definition is manipulated using the Linked Attribute Database Definition screen.

When a "linked" attribute file is initially attached, a new dBase IV format database file possessing the same name as the model (with a ".cdb" extension for customer data, ".ndb" extension for node data, or a ".pdb" extension for pipe data) is created. A basic Attribute Database Definition is used to create the new file. Additional fields may be added to the file to meet the User's preferences and requirements.

An Attribute Database Definition is modified using the Linked Attribute Database Definition screen. The screen may be accessed by clicking the *Edit Attribute Database Definition* command button on the associated Attribute File Settings screen. The features associated with the screen are described as follows.

Displayed Items

Field No - Displays the position of the "field" in the database. This value is automatically assigned and cannot be changed by the User.

Data Items

Allow Edit - Indicates whether the data "field" values may be edited using the various Display, Edit, and Report routines.

Data List - Specifies the name of the selection list to display when a data "field" is being edited using one of the Display, Edit, and Report routines.

Display - Indicates whether the data "field" will be displayed in the various Display and Report routines.

Display Text - Specifies the text value to display when a data "field" is shown using the various Display, Edit, and Report.

Field Name - Specifies the name for the data "field." See the following *Notes & Considerations* section for the *Field Name* requirements.

Field Type - Specifies the data type the data "field" will store. Text and a variety of numeric types are supported.

Field Width - Specifies the maximum number of characters in the data "field."

Icons

 Add Field - Adds a new field to the end of the definition.
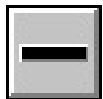
 Delete Field - Deletes the selected (highlighted) field.

 Delete Fields - Deletes all of the non-required fields. The required fields (LINK_ID and ADDRESS) will remain.
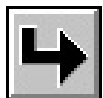
 Insert Field - Inserts a new field just before the selected (highlighted) field.
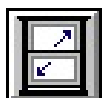
 Maximize Screen Size - Increases the Linked Attribute Database Definition screen size to fill the GASWorkS display.

Move Field - Moves the selected (highlighted) field to a new location in the definition. When clicked, a message will be displayed to select the new location. Click the *OK* command button to clear the message, then select the desired location.

Open - Opens a previously saved Attribute Database Definition file (.ddf). When clicked, the File Selection screen is displayed.

Print - Prints the contents of the definition. When clicked, the Print Settings screen is displayed.
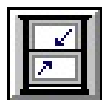
Reset Screen Size - Sets the Linked Attribute Database Definition screen size to its "normal" dimensions and position.
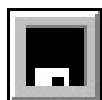
Save - Saves the contents of the currently displayed definition. When clicked, the File Selection screen is displayed. Once saved, the file may be opened and used with other models.

Command Buttons

Apply - Closes the screen and applies changes.

Cancel - Closes the screen without saving changes.

Notes & Considerations

● Data cells with a "yellow" background color display assigned or calculated values. Values in these cells cannot be directly changed by the User.

● Each row in the Attribute Database Definition represents a field in the attribute data file. The fields are displayed in the order that they appear in the definition.

● No more than 255 fields may be included in an Attribute Database Definition.

● The *Field Name* item must be less than ten (10) characters. The Field Names cannot contain spaces or special characters (like %, #, !). Use an underscore (_) character instead of a blank space in a Field Name.

● An outside application, such as Microsoft Excel, may be used to edit the attribute data file. However, doing so may corrupt the Attribute Database Definition used by GASWorkS and may result in a loss of data or connection between GASWorkS and the attribute data file.

● Additional fields may be added or removed from the attribute data file after its initial creation by revising the Attribute Database Definition. Revising the Attribute Database Definition will require the attribute data file to be recreated. When the file is recreated, if a field is removed or a *Field Name* or *Field Type* is changed, some data may be lost. If a change results in an unexpected loss of a data field, close the Attribute Database Definition without saving the changes, and the old definition and values will be preserved.

● When an attribute file (database) is attached, the model and database attribute files are linked using the Link ID Number for customers and pipes and the Name or Internal ID Number for nodes. When using a "linked" style database, the Link ID Number is automatically managed by GASWorkS. Do not change the Link ID Number of a feature or database record when using a "linked" style attribute file.

● When a new "linked" attribute data file is first attached, a basic Attribute Database Definition will be created. The initial definition will contain a field named "LINK_ID" and one named "ADDRESS." Additional fields may be added, or if a definition has been previously saved, it may be opened and applied to the current model.

● If a "linked" style attribute data file is to be used, the file should be attached, and the Attribute Database Definition should be created immediately after creating a new model, or the New Project routine should be used to specify the Attribute Database Definition and automatically attach the attribute data file while creating the new model.

● The LINK_ID and ADDRESS fields are required by the customer, node, and pipe attribute data files. Do not remove these fields from the Attribute Database Definition. Their display may be turned "off" ("No") in the definition if you do not wish to view them.

● Several sample Attribute Database Definitions are included when GASWorkS is installed. The files all end in a ".ddf" extension and are located in the "support\definitions" subdirectory (subfolder) of the root GASWorkS directory (folder). A description of the included files is found in the *Database Definition Files* topic in the *Appendix* section.

Attribute File Settings Screen

An attribute (non-model) database attribute file may be attached to customer, node, and pipe features in a model. The attribute file settings are defined using the associated Attribute File Settings screen. The screen may be accessed by selecting the *Attribute File* item from the *Customer Data*, *Pipe Data*, or *Node Data* menu of the *Utilities* menu. The features associated with the screen are described as follows.

General Tab

Data Items

Address Field - Specifies the name of the database "field" (in the attached attribute file), which contains the associated address values. The Address Field is used for certain Report routines. This item is optional; if unused, leave the list empty. This item is only enabled for certain attribute file types.

Attribute File Type - Specifies the type of attached attribute file.

Connection Status - Specifies the connection status between the model and attribute file. If the status is turned "off," the connection will be disabled until turned back "on."

Filename - Specifies the name of the attached attribute file. Enter a Filename or click the *Browse* command button to select a Filename. This item is only enabled for non-linked attribute file types.

Link ID Number Field - Specifies the name of the database "field" (in the attached attribute file) which contains the Link ID (identification) Number. The Link ID Number is used to "link" the associated features contained in the model to the data records contained in the attached attribute file. This item is only enabled for non-linked attribute file types.

Total Load Field - Specifies the name of the database "field" (in the attached attribute file) which contains the *Total Load* values. If this item is specified, it is assigned to the *Total Load* value of the associated feature. This item is optional, if it is not used leave the list empty. This item is only enabled for customer attribute file types.

Total Load Units - Specifies the dimensional units for the *Total Load* values in the attached attribute file. If the *Total Load Field* is specified, this item must be assigned. This item is only enabled for customer attribute file types.

Use The Model Path Location For The Attribute File Location - When selected, the same location as the associated model will be used for the attribute file. This item is useful when models are frequently shared or moved. It eliminates the need to reset the attribute file location each time the model is moved. This item is only enabled for non-linked attribute file types.

Other Tab

The *Other* data tab will only be enabled for the *Attribute File Types* other than "None."

Data Items

Attribute Database Definition - Displays the data "fields" in the current Attribute Database Definition. The list only allows the contents of the Attribute Database Definition to be viewed. Click the *Edit Attribute Database Definition* command button to edit or revise a "linked" Attribute Database Definition.

Command Buttons

Browse - Used to find and select an attribute database file name. When executed, displays the File Selection screen. This command button is displayed on the *General* data tab.

Cancel - Closes the screen without saving changes.

Close - Closes the screen, saves changes, and attaches the attribute file.

GASWorkS™ 10.1   *Attribute Data*

Edit Attribute Database Definition - Displays the Linked Attribute Database Definition screen to edit the contents. This command button is displayed on the *Other* data tab and is only enabled when a Linked Attribute *Attribute File Type* is attached.

Notes & Considerations

● Data cells with a "yellow" background color display assigned or calculated values. Values in these cells cannot be directly changed by the User.

● The available *Attribute File Types* are described in the following table:

File Type	Description
CSV (Comma-Separated Values)	Select this item when the attribute data is in a Comma-Separated Values (CSV) text format. The file does not need to contain a record for each associated feature. Only records that match an existing feature will be available for display. Features are matched using the *Link ID Number* item.
dBase III	Select this item when the attribute data is in a dBase III format. The file does not need to contain a record for each associated feature. Only records that match an existing feature will be available for display. Features are matched using the *Link ID Number* item.
dBase IV	Select this item when the attribute data is in a dBase IV format. The file does not need to contain a record for each associated feature. Only records that match an existing feature will be available for display. Features are matched using the *Link ID Number* item.
Excel 8.0 (XLS Workbook)	Select this item when the attribute data is in an Excel 8.0 Single-Sheet Workbook format file. The file does not need to contain a record for each associated feature. Only records that match an existing feature will be available for display. Features are matched using the *Link ID Number* item.
Linked Attribute	Select this item to use the GASWorkS "linked" attribute file features. When this type of file is selected, GASWorkS creates a database file based on a User-specified definition and manages the addition and deletion of records associated with the file. See the notes on the Database Definition item if using this database type.
None	Use this item when no attribute file is attached.

● When an attribute file is attached and the *Connection Status* is turned "on," the attribute data will be displayed by the various Display, Edit, and Report routines. If a "linked" database is attached, the contents may be available for manipulation with the various "*Edit*" commands and the SQL edit feature.

● After attaching or changing a customer attribute file, remember to update the node loads, as appropriate, using the *Update External Loads* item from the *Customer Data* submenu of the *Utilities* menu.

● When a linked attribute file is initially attached, GASWorkS creates a new dBase IV (4.0) format database file possessing the same name as the model, with a ".cdb," ".ndb," or ".pdb" extension. The contents of the file are specified by the Attribute Database Definition. Once created, additional fields may be added or removed from a linked attribute file by revising the Attribute Database Definition. See the *Linked Attribute Database Definition Screen* topic in this section of this manual.

● Once a linked attribute file is attached and created, GASWorkS manages the addition and deletion of the associated records. One database record is maintained for each data feature contained in the model. Individual features may be "attached" or "unattached" from the attribute file by appropriately setting the *Link To Database* item.

● The contents of a linked attribute file may be manipulated by an outside application. However, care should be taken so that the *Link ID Number* item and its values are not corrupted. If changes are made by an outside application, ensure that GASWorkS is not using the file when the changes are made. Do not add or remove records from the file using a non-GASWorkS application.

● When an attribute file (database) is attached, the model and database attribute files are linked using the Link ID Number for customers and pipes and the Name or Internal ID Number for nodes. When using a "linked" style database, the Link ID Number is automatically managed by GASWorkS. Do <u>not</u> change the Link ID Number of a feature or database record when using a "linked" style attribute file.

● When an attribute file (database) is attached, the model and database files are linked using the *Link ID Number* item. When a function needs to display or report information contained in the attribute file, a search of the database is performed. The first record in the database with a matching Link ID Number is retrieved. Multiple features may contain the same Link ID Number. However, unique Link ID Numbers should appear in the database.

● Depending on the size of the attached database, processor speed, and "connection" speed, the search process may take some time to complete. If the connection is slow, and the information in the attribute file is not required, the attribute file *Connection Status* may be temporarily turned "off" when the attribute data is not needed.

● When working with a linked attribute file, turning the *Connection Status* to "off" does not prevent records from being added to or deleted from the database file - it only suppresses the display of the attribute data in the various Display and Report routines.

● Each database style has unique requirements for Field Name, size, and syntax. In general, keep the Field Names short (less than ten (10) characters), do not include spaces in the name (use an underscore (_) character instead), and do not use special characters (like %, #, !, etc.) in the name.

● When a data list is used with a linked attribute file, the assigned list will be displayed when the associated data "field" is edited. See the *Using Data List* topic found later in this section of the manual for more information.

Update External Loads

The node data contains an item known as the external load. This value refers to loads assigned from an external source using the Update External Loads routine. The routine is found in the *Customer Data* submenu of the *Utilities* menu.

The routine behaves differently depending on whether a customer attribute file (database) is attached and turned "on." If the database Connection Status is "on" and the *Total Load* value is assigned, the routine looks for a matching record in the database, based on the Link ID Number, for each customer feature in the model. If a match is made, the customer load is updated with the value found in the attribute file. When all customer loads have been updated, the node external loads are updated with the new customer loads based on the *Node Load Application* method assigned to each customer. For customers whose *Adjust Load* option is "Yes," the load will be assigned to the calculated *External Load* item. Where the *Adjust Load* option is "No," the load will be assigned to the calculated *External Load (Fixed)* item. If the database Connection Status is "off" or nondatabase is attached, a similar process is performed. However, the customer load values are not updated. Only the external load values for the nodes are updated.

Using Data Lists

When a linked attribute file is used, the data "fields" contained in the file may be specified by the User. To make data entry for these fields more efficient, a data list may be assigned to a field. Selecting a value from a data list is generally more efficient than entering a value by typing it into a data field. Data "fields" that represent items that may be repeated many times in the database (like customer type or meter size/type) are good candidates to associate with a data list. Generally, data "fields" that represent a number or text value that may be unique or not used often are not good candidates to associate with data lists.

A data list is a simple ASCII text file that contains a list of data items and values. The format of the file is as follows:

> Data Item
> Item Value
> Data Item
> Item Value
> etc...

The "Data Item" represents the text or value to be displayed in the data list, and the "Item Value" represents a value to associate with the Data Item. The usage of the Item Value will be explained later in this section. Item Values can only be integer-type numbers ranging from -32000 to 32000. An Item Value may be zero (0), but a value <u>must</u> be entered.

An existing data list may be edited, or a new one may be created using a *Data List Definition* item in the *Edit* menu. After a data list has been created, it may be assigned to an associated data "field" using the *Attribute File* item from one of the *Customer Data*, *Node Data*, or *Pipe Data* submenus of the *Utilities* menu. Once assigned, the data list will be displayed when the associated item is edited. The first item in the list will be used as the default setting for the new features.

The storage and memory required to store an item in the database may be reduced by using a numeric integer code to represent the Data Item instead of its actual text value. For example, pipe coating types might be coded so that "1" represents "Polyethylene Plastic" and "2" represents "Fusion Bonded Epoxy." The storage requirement for saving the code "1" is much less than storing the entire text string "Polyethylene Plastic" (2 bytes to store the integer code compared to 20 bytes to store the text string).

GASWorkS accommodates using item codes through the Item Value in the data list. When a Field Type is set to an "Integer - Short" Field Type and a data list is associated with that field, the Data Item values are displayed when the field is edited by one of the Edit or Report routines; however, when the item is saved, the code specified by the associated Item Value is saved in the database. This allows a more efficient but slightly more complicated method of storing text items that are appropriate for inclusion in a data list.

There are two special types of data list files. One indicates that the contents of the selection list should be based on a Property Table, the other indicates that the contents of the selection list should be based on a GASWorkS dimensional units list.

To create a data list that specifies that a Property Table should be used, a file that contains one record must be created. The "Item Description" must contain the value **GW_PROPERTY_TABLE**. The "Item Value" must contain the name of the Property Table to use. Acceptable values for the "Item Value" are:

COMPRESSOR	**FITTING**
PIPE	**REGULATOR**
VALVE	**WELL**

When the data list is displayed, it will contain the contents of the currently assigned Property Table, as specified by the Property Table Selection screen.

To create a data list that specifies that a dimensional units list should be used, a file that contains one record must be created. The "Item Description" must contain the value **GW_UNITS_LIST**. The "Item Value" must contain the name of the unit type to use. Acceptable values for the "Item Value" are:

COORDINATE	**LENGTH**
DIAMETER	**POWER**
EFFICIENCY	**PRESSURE**
ELEVATION	**TEMPERATURE**
ENERGY	**VELOCITY**
FLOW	**VISCOSITY**
FUEL	**VOLUME**
HEATING	

When the data list is displayed, it will contain the contents of the specified dimensional units list, as predefined in the GASWorkS software.

A comment or remark may be included in a data list file by beginning the line with a pair of forward slashes (//). The comment must immediately follow on the same line. Comments are not included when the list is displayed in a data field.

Several sample data lists are included when GASWorkS is installed. These files can be opened and viewed, for example to arrange a data list file. The files all end in a ".lst" extension and are located in the "support\datalists" subdirectory (subfolder) of the root GASWorkS directory (folder). A description of the files may be found in the *Data List Files* topic in the *Appendix* section.

Using SQL Statements

GASWorkS provides a routine for updating data values in the linked attribute database. This routine is useful for modifying customer load data in an attribute file (database). The modifications are defined by using a Standard Query Language (SQL) statement. SQL is a special language used to control and define changes to supported format database files. A complete discussion and review of SQL Statements is beyond the scope of this document. The SQL Statement is entered and executed using the SQL Update Specifications screen. The features associated with the screen are described as follows.

SQL Update Specifications Screen

Displayed Items

Field Reference - Displays the association between the displayed field descriptions and the field name. SQL Statements require the use of actual *Field Name* values.

Data Items

Saved SQL Statements - Specifies whether a previously saved SQL Statement will be used for the current SQL statement.

SQL Statement - Specifies the desired SQL statement. See the following *Notes & Considerations* section for further information about the statement format requirements.

Command Buttons

Close - Closes the screen and saves changes without proceeding with the Update routine.

Delete Displayed Statement - Deletes the statement currently displayed in the *Saved SQL Statements* list.

Execute - Executes the specified SQL Statement.

Execute & Update - Executes the current SQL Statement and updates the model accordingly.

Save Current Statement - Saves the currently displayed SQL Statement.

Notes & Considerations

● Any previously saved SQL Statements will be displayed in the *Saved SQL Statements* list.

● The SQL Statement represents the "Set" and "Where" portions of a standard update statement. For example, to increase the values in a field called "TOTAL" by ten percent (10%), the required SQL Statement would be: SET TOTAL = TOTAL * 1.1.

● Changes made to the attribute file using this method are "permanent." You may be able to undo a change by immediately executing a "reciprocal" SQL Statement. For example, to undo the change described above, the reciprocal SQL Statement would be: SET TOTAL = TOTAL / 1.1. The GASWorkS *Undo* command does not restore changes made using a SQL Statement.

IMPORT, MERGE, & PURGE ROUTINES

The import and merge routines are similar because they can create model features from data saved in another format. The difference in the routines is that import routines create a new model from outside data, and the merge routine adds data from an outside source to an existing model. Different data sources are handled using specific routines and specifications. The specific requirements of each routine are described in the following sections.

Import Routine

This section provides descriptions and requirements for importing data from the various supported file formats. The additional data screens used to further define the import specifications will be described later in this section. See the Help Guide for step-by-step examples of importing various data file types.

GASWorkS provides translation routines for importing data associated with several file formats. Using the Import routine, data contained in ASCII Text files, older version GASWorkS files, database files, spreadsheet files, CAD files, and GIS files may be used with GASWorkS 10. The following table describes the Import *File Types* supported by GASWorkS.

File Type
CSV (Comma-Separated Values) File - Customer, Pipe
DXF (Drawing Exchange Format) File
GASWorkS 7.0, 8.0, 9.0
KML (Keyhole Markup Language) File
SHP (ESRI Shapefile) File - Customer, Pipe
Stoner PD/XY File

The Import routine is accessed through the *Import* item of the *Utilities* menu. When selected, the Import screen is displayed. The features associated with the Import screen, along with the specific requirements associated with each supported file type, are described in the following sections.

When importing data files, ensure that the data values and dimensional units in the current file are compatible with the values supported by GASWorkS. The import file does not need to be completely inclusive. Items not included in the file will be set to applicable default data values. The default data values may be set before importing a file using the *Set Default Data Values* command button on the Import screen. Feature data may be revised after the Import routine is complete using the various Mass Update and Edit routines.

Import Screen
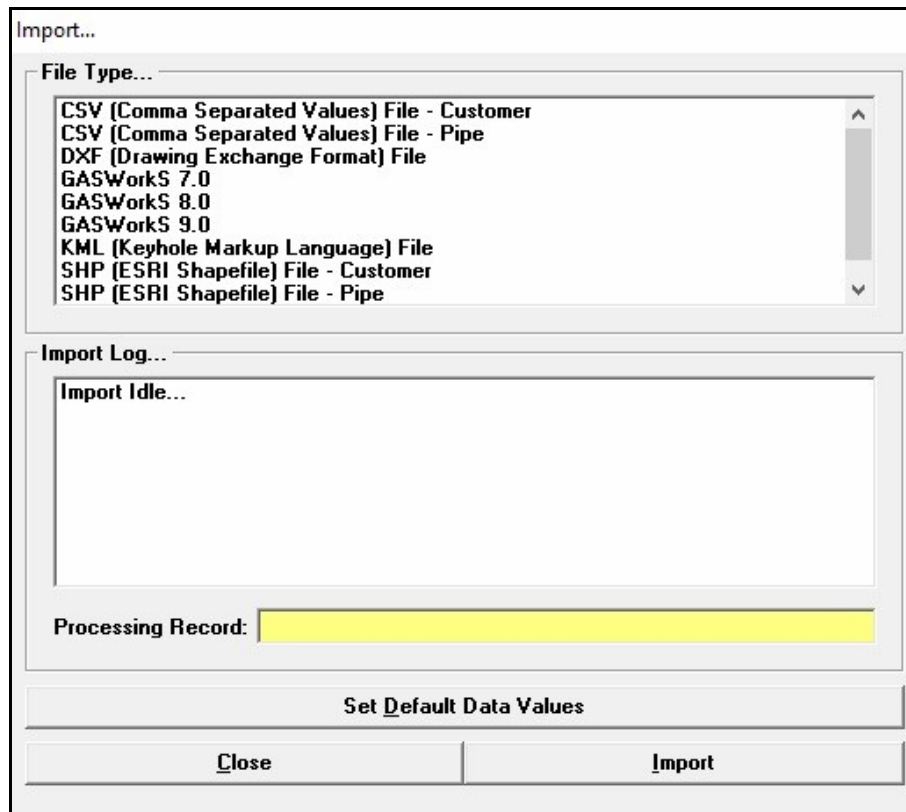
Displayed Items

Import Log - Displays the activity (status) of the Import routine.

Processing Record - Displays the number of the record currently being processed or the number of the records that were processed when completed.

Data Items

File Type - Displays the file types that may be imported.

Command Buttons

Close - Closes the screen.

Import - Starts the Import routine. When clicked, the File Selection screen is displayed.

Set Default Data Values - Allows the Default Data Values to be set. When clicked, the Default Data Values screen is displayed.

Notes & Considerations

● The Import routine creates a new set of GASWorkS model data files. If existing model files are present, they will be deleted and sent to the Windows Recycle Bin before the Import routine is executed.

● Files opened by another application, like a database or spreadsheet application, cannot be processed by the Import routine until they are closed by the outside application.

● The Merge routine may add data to an existing model.

● The Purge routine may remove items from an existing model.

Merge Routine

This section provides descriptions and requirements for merging data from the various supported file formats. The additional data screens used to further define the merge specifications will be described later in this section. See the Help Guide for step-by-step examples of merging various data file types.

GASWorkS provides translation routines for merging data associated with several file formats. Using the Merge routine, data contained in older version GASWorkS files, database files, spreadsheet files, CAD files, and GIS files may be added to an existing GASWorkS 10 model. The following table describes the Merge *File Types* supported by GASWorkS.

File Type
CSV (Comma-Separated Values) File - Customer, Node
DXF (Drawing Exchange Format) File
KML (Keyhole Markup Language) File
SHP (ESRI Shapefile) File - Customer, Junction, Node, Pipe, Service, Valve

The Merge routine is accessed through the *Merge* item of the *Utilities* menu. When selected, the Merge screen is displayed. The features associated with the Merge screen, along with the specific requirements associated with each supported file type, are described in the following sections.

When merging data files, ensure that the data values and dimensional units in the current file are compatible with the values supported by GASWorkS. The merge file does not need to be completely inclusive. Items not included in the file will be set to applicable default data values and may be revised after the Merge routine is complete, using either the Mass Update or individual Edit routines. The default data values may be set before merging a file using the *Set Default Data Values* command button on the Merge screen.

Merge Screen
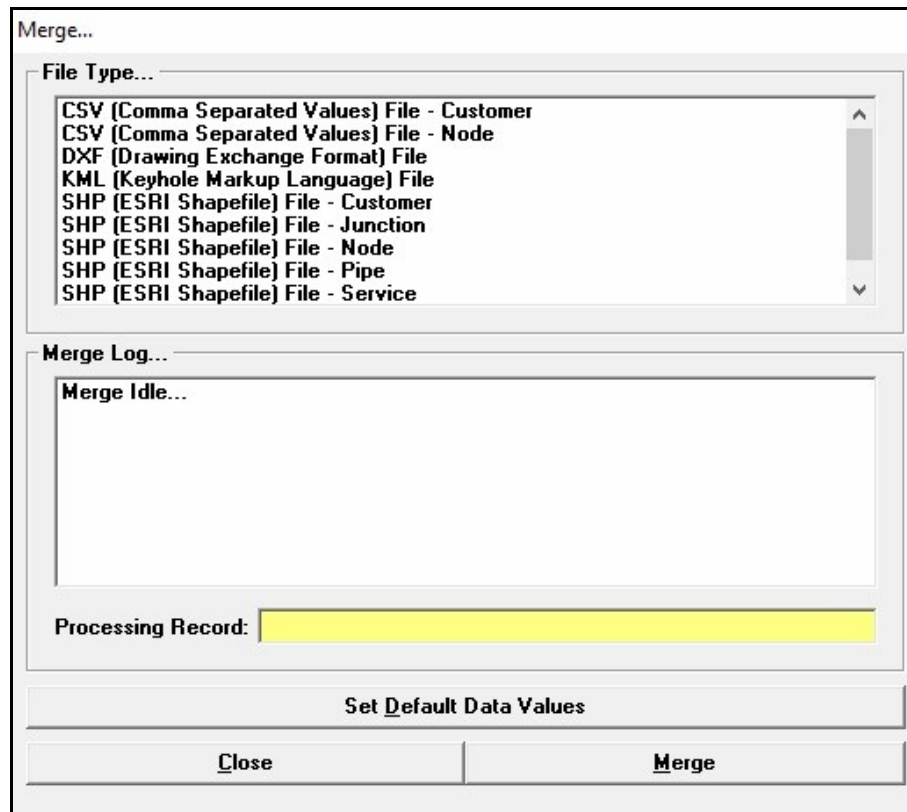
Displayed Items

Merge Log - Displays the activity (status) of the Merge routine.

Processing Record - Displays the number of the record currently being processed or the number of the records that were processed when completed.

Data Items

File Type - Displays the type of file to be merged.

Command Buttons

Close - Closes the screen.

Merge - Starts the Merge routine. When selected, displays the File Selection screen.

Set Default Data Values - Allows the Default Data Values to be set. When clicked, the Default Data Values screen is displayed.

Notes & Considerations

● The Merge routine allows a file to be merged with the currently open model. When using the routine, the selected merge (source) file is compared to the model. Where matching records are found, only the defined data fields are updated. If a matching record is not found in the GASWorkS file, a new record is added to it.

● Depending on the current file type, additional specification screens may be displayed. If a screen appears, enter the appropriate data and settings to continue.

Import Or Merge CSV (Comma-Separated Values) Files

When importing or merging a CSV (Comma-Separated Values) file, certain information must be specified about the selected file. The specifications and settings for importing or merging a CSV file are defined using the associated Import or Merge Specifications screen. The screens are displayed when a *CSV (Comma Separated Values) File* type item is selected during the Import or Merge routines. The same specification screen is used for the Import and Merge routines; some items will not be enabled depending on the routine being used. The screen description follows:

Customer Import Or Merge Specifications Screen - *General Settings*

GASWorkS™ 10.1   *Import, Merge, & Purge*

Displayed Items

Model Coordinate Units - Displays the dimensional units for the coordinate values in the model file. It is used to make the appropriate conversion between lengths, source coordinate units, and model Coordinate units. Click the *Set Default Values* command button to change the model Coordinate units.

Data Items

Data Field Assignments - Displays the association between fields in the CSV file and data items in GASWorkS. To make an assignment - in the *CSV Column Assignment* column, click on a cell and select an item from the list; in the adjacent cell, in the *GASWorkS Data Item* column, select an item from the list.

CSV Column Assignment - Displays and allows the selection of fields from the CSV file.

GASWorkS Data Item - Displays and allows the selection of GASWorkS data items.

Origin Shift - Specifies the amount to shift the source coordinate values during the Import or Merge routine. The X-value indicates the horizontal (east-west) amount to shift the coordinates. The Y-value indicates the vertical (north-south) amount to shift the coordinates. The values are expressed in model Coordinate units.

Scale Factor - Specifies the value to scale the source coordinate values by.

Source File Coordinate Units - Specifies the dimensional units for the coordinate values in the source file. This item is used to make the appropriate conversion between coordinate values contained in the source file and the model. Not currently used.

Customer Import Or Merge Specifications Screen - *Other Settings*

Data Items

Assign Service Lines During Import/Merge - When selected, the "service lines" will be created using the selected *Assignment Method* during the Import or Merge routine.

Assignment Method - Specifies the supply main selection method when service lines are assigned during the Import or Merge routine. See the *Notes & Considerations* section for a description of the assignment methods.

Fuzzy Tolerance - Specifies the *Fuzzy Tolerance* value when processing the source file. Only displayed using the Merge routine and when the "By Coordinate Location" option is selected for the Search Method.

Load Assignment Method When Match Found - Specifies the method to be used when a matching customer location is found during the Merge routine. See the *Notes & Considerations* section for a description of the assignment methods.

Maximum Service Line Length - Specifies the maximum search distance using the "Nearest Main" assignment method. The value is expressed in model Coordinate units.

Multiply Load Value By (-1) During Import/Merge - When selected, if a load value is assigned, the value will be multiplied by -1 during the Import or Merge routine.

Search Method - Specifies the method used to search for a "matching" (existing) customer during the Merge routine. See the *Notes & Considerations* section for a description of the search methods.

Command Buttons

Cancel - Closes the screen without saving changes to the specification or proceeding with the associated routine.

Clear Data Field Assignments - Clears all of the Data Field Assignments. This command button is displayed on the *General Settings* data tab.

Close - Closes the screen, saves changes to the specification, and does not proceed with the associated routine.

Continue - Saves the current specification settings and proceeds with the associated routine.

Open Specification - Opens a previously saved specifications file (.cxi). When clicked, the File Selection screen is displayed.

Save Specification - Saves the current specification values. When clicked, the File Selection screen is displayed.

Set Default Values - Allows changes to the default data values. When clicked, the Default Data Values screen is displayed.

Notes & Considerations

● Do not leave empty rows between entries in the *Data Field Assignments* list.

● When merging a customer data file into an existing model, search options are provided to determine if a customer in the data file already exists in the model. If it does, the *Load Per Unit* value of the matching customer is updated according to the specified *Load Assignment Method When Match Found*. If a match is not found, a new customer feature is not created in the model, and its data values are assigned accordingly. The search options and their behavior are listed below:

> *None - Do Not Search For Match* - When selected, no search is performed. When this option is used, a customer will be placed regardless of whether a customer already exists at the same location.

> *By Coordinate Location* - When selected, a search is performed to determine if the customer location from the data file falls within the specified Fuzzy Tolerance of an existing customer in the model. If it does, it is considered to be a "match."

> *By Link ID Number* - When selected, a search is performed to determine if the Link ID Number associated with the customer from the data file is already assigned to an existing customer in the model. If it does, it is considered to be a "match."

● When merging a customer data file into an existing model using the "By Coordinate Location" or "By Link ID Number" search options, when a match is found, load values from the data file are assigned according to the selected *Load Assignment Method When Match Found* method. The assignment methods and their behavior are listed below:

> *Add To Existing Value* - When selected, when a match is found, the value from the data field assigned to the *Load Per Unit* GASWorkS Data Item will be added to the current *Per Unit Load* value for the matching customer in the model.

> *Do Not Change Existing Value* - When selected, when a match is found, no action is taken. The *Per Unit Load* value for the matching customer in the model is unchanged.

> *Replace Existing Value* - When selected, when a match is found, the current *Per Unit Load* value for the matching customer in the model will be replaced with the value from the data field assigned to the *Load Per Unit* GASWorkS Data Item.

> *Use Larger Value* - When selected, when a match is found, the current *Per Unit Load* value for the matching customer in the model will be compared to the value from the data field assigned to the *Load Per Unit* GASWorkS Data Item. The current *Per Unit Load* value for the matching customer in the model will be set to the larger of the two values.

> *Note* - If no data field is assigned to the *Load Per Unit* GASWorkS Data Item, the current *Per Unit Load* value for the matching customer in the model will not be changed.

● When coordinate values are assigned in the *Data Field Assignments* list, the values read from the data file will be changed as follows:

    ● The coordinate values will first be multiplied by the specified Scale Factor value.

    ● The scaled coordinate values will then be increased or decreased by the values specified for the Origin Shift values.

Node Merge Specifications Screen - *Other Settings*

```
Node Merge Specifications...

┌─────────────────────┐ ┌──────────────────────┐
│  General Settings   │ │    Other Settings    │
└─────────────────────┘ └──────────────────────┘

┌─ Existing Node Matching Options... ──────────────────┐
│                                                       │
│  Search Method:  By Node Name                    ▼    │
│                                                       │
│  Fuzzy Tolerance (Model Length Units):    1      ▼    │
│                                                       │
│  Load Assignment Method When Match Found:             │
│                             Replace Existing Value ▼  │
└───────────────────────────────────────────────────────┘

┌─ Other Options... ───────────────────────────────────┐
│                                                       │
│   ☐ Multiply Load Value By (-1) During Merge          │
│                                                       │
│                                                       │
│                                                       │
│                                                       │
│                                                       │
│                                                       │
│                                                       │
└───────────────────────────────────────────────────────┘

┌──────────────────┬──────────────────┬──────────────────┐
│ Set Default Values│ Save Specification│ Open Specification│
├──────────────────┼──────────────────┼──────────────────┤
│      Cancel      │      Close       │    Continue...   │
└──────────────────┴──────────────────┴──────────────────┘
```

Data Items

Fuzzy Tolerance - Specifies the *Fuzzy Tolerance* value when processing the source file. Only displayed when the "By Coordinate Location" option is selected for the *Search Method*.

Load Assignment Method When Match Found - Specifies the method to be used when a matching customer location is found during the Merge routine. See the *Notes & Considerations* section for a description of the assignment methods.

Multiply Load Value By (-1) During Merge - When selected, if a load value is assigned, the value will be multiplied by -1 during the Merge routine.

Search Method - Specifies the method used to search for a "matching" (existing) node during the Merge routine. See the *Notes & Considerations* section for a description of the search methods.

Command Buttons

Cancel - Closes the screen without saving changes to the specification or proceeding with the associated routine.

Clear Data Field Assignments - Clears all of the Data Field Assignments. This command button is displayed on the *General Settings* data tab.

Close - Closes the screen, saves changes to the specification, and does not proceed with the associated routine.

Continue - Saves the current specification settings and proceeds with the associated routine.

Open Specification - Opens a previously saved specifications file (.nxi). When clicked, the File Selection screen is displayed.

Save Specification - Saves the current specification values. When clicked, the File Selection screen is displayed.

Set Default Values - Allows changes to the default data values. When clicked, the Default Data Values screen is displayed.

Notes & Considerations

● The most common use of the node merge routine is to update load values in an existing model when certain nodes represent customer demand locations or to update input flow rates when certain nodes represent supply points like production wells.

● Do not leave empty rows between entries in the *Data Field Assignments* list.

● When merging a node data file into an existing model, search options are provided to determine if a node in the data file already exists in the model. If it does, the *Base Load* value of the matching node is updated according to the specified *Load Assignment Method When Match Found*. If a match is not found, the unmatched node in the data file is ignored. The search options and their behavior are listed below.

> *By Coordinate Location* - When selected, a search is performed to determine if the node location from the data file falls within the specified Fuzzy Tolerance of an existing node in the model. If it does, it is considered to be a "match."

> *By Node Name* - When selected, a search is performed to determine if the *Name* value associated with the node from the data file is already assigned to an existing node in the model. If it does, it is considered to be a "match."

● When merging a node data file into an existing model using the "By Coordinate Location" or "By Node Name" search options, when a match is found, load values from the data file are assigned according to the selected *Load Assignment Method When Match Found* method. The assignment methods and their behavior are listed below.

> *Add To Existing Value* - When selected, when a match is found, the value from the data field assigned to the *Load, Base* GASWorkS Data Item, will be added to the current *Base Load* value for the matching node in the model.

> *Replace Existing Value* - When selected, when a match is found, the current *Base Load* value for the matching node in the model will be replaced with the value from the data field assigned to the *Load, Base* GASWorkS Data Item.

> *Use Larger Value* - When selected, when a match is found, the current *Base Load* value for the matching node in the model will be compared to the value from the data field assigned to the *Load, Base* GASWorkS Data Item. The current *Base Load* value for the matching node in the model will be set to the larger of the two values.

> *Note* - If no data field is assigned to the *Load, Base* GASWorkS Data Item, the current *Base Load* value for the matching node in the model will not be changed.

● When coordinate values are assigned in the *Data Field Assignments* list, the values read from the data file will be changed as follows:

> • The coordinate values will first be multiplied by the specified Scale Factor value.

> • The scaled coordinate values will then be increased or decreased by the values specified for the Origin Shift values.

GASWorkS™ 10.1 — *Import, Merge, & Purge*

Pipe Import Specifications Screen

Displayed Items

Model Coordinate Units - Displays the dimensional units for the coordinate values in the model file. It is used to make the appropriate conversion between lengths, source coordinate units, and model Coordinate units. Click the *Set Default Values* command button to change the model Coordinate units.

Data Items

Data Field Assignments - Displays the association between fields in the CSV file and data items in GASWorkS. To make an assignment - in the *Database Field Item* column, click in a cell and select an item from the list; in the adjacent cell, in the *GASWorkS Data Item* column, select an item from the list.

        Database Field Item- Displays and allows selection of fields from the CSV file.

        GASWorkS Data Item - Displays and allows selection of GASWorkS data items.

Fuzzy Tolerance - Specifies the *Fuzzy Tolerance* value when processing the source file. During the Import routine, line ends that fall within the *Fuzzy Tolerance* value are snapped together.

Origin Shift - Specifies the amount to shift the source coordinate values during the Import routine. The X-value indicates the horizontal (east-west) amount to shift the coordinates. The Y-value indicates the vertical (north-south) amount to shift the coordinates. The values are expressed in model Coordinate units.

Scale Factor - Specifies the value to scale the source coordinate values by.

Source File Coordinate Units - Specifies the dimensional units for the coordinate values in the source file. This item is used to make the appropriate conversion between coordinate values contained in the source file and the model. Not currently used.

Command Buttons

Cancel - Closes the screen without saving changes to the specification or proceeding with the associated routine.

Clear Data Field Assignments - Clears all of the Data Field Assignments. This command button is displayed on the *General Settings* data tab.

Close - Closes the screen, saves changes to the specification, and does not proceed with the associated routine.

Continue - Saves the current specification settings and proceeds with the associated routine.

Open Specification - Opens a previously saved specifications file (.sxi). When clicked, the File Selection screen is displayed.

Save Specification - Saves the current specification values. When clicked, the File Selection screen is displayed.

Set Default Values - Allows changes to the default data values. When clicked, the Default Data Values screen is displayed.

Notes & Considerations

● Do not leave empty rows between entries in the *Data Field Assignments* list.

● When coordinate values are assigned in the *Data Field Assignments* list, the values read from the data file will be changed as follows:

> • The coordinate values will first be multiplied by the specified Scale Factor value.

> • The scaled coordinate values will then be increased or decreased by the values specified for the Origin Shift values.

● During the Import routine, pipe ends that fall within the *Fuzzy Tolerance* value are considered a "match." When a matching pipe is found in the model, only the assigned data values are updated. When no match is found, a new pipe feature is added to the model.

● Only "two-point" pipe types (pipes with no vertices) can be imported using the CSV file format.

GASWorkS™ 10.1   *Import, Merge, & Purge*

Import Or Merge DXF (Drawing Exchange Format) Files

When importing or merging a DXF (Drawing Exchange Format) file, certain information must be specified about the selected file. The import or merge specification and settings for a DXF file are defined using the Import Or Merge Specifications screen. The screen is displayed when the *DXF (Drawing Exchange Format) File* File Type is selected from the Import or Merge screen. The features associated with the screen are described as follows.

Import Or Merge Specifications Screen - *Pipe Settings*

Data Items

Arc Resolution - Specifies the resolution when processing an arc from the source file. If the value is greater than zero (0), arc entities will be converted to polyline-type pipes with a vertex placed according to the *Arc Resolution* value. The value is expressed in terms of the degrees of arc.

Assign Entity Handle To Pipe Link ID Number - When selected, the entity handles in the drawing are assigned to the Link ID Number field for pipes processed during the Import or Merge routine.

Fuzzy Tolerance - Specifies the *Fuzzy Tolerance* value when processing the source file. Line ends that fall within the *Fuzzy Tolerance* value are snapped together.

Pipe Layers - Displays the Layer Assignment and Pipe Size assignments. To make an assignment - under the *Layer Assignment* column, click on the associated cell and select an item from the list. Click in the adjacent cell under the *Pipe Size* column and select an item from the list to complete the assignment.

>   **Layer Assignment -** Displays the selected fields from the DXF file.

>   **Pipe Size -** Displays the selected GASWorkS size value.

Pipe Length Units - Specifies the length dimensional units for the imported or merged pipes. It is used to make the appropriate conversion between pipe lengths, drawing coordinate units, and the model Coordinate units.

Valve Node Layer - Specifies whether valve nodes will be processed during the Import or Merge routine. If symbols (blocks) indicating the location of the valves are present in the drawing, select the associated layer. If a layer is assigned, blocks will be processed during the Import or Merge routine.

GASWorkS™ 10.1   *Import, Merge, & Purge*

Import Or Merge Specifications Screen - *Other Settings*

Displayed Items

Model Coordinate Units - Displays the dimensional units for the coordinate values in the model file. It is used to make the appropriate conversion between lengths, source coordinate units, and model Coordinate units. Click the *Set Default Values* command button to change the model Coordinate units.

Data Items

Allow Multiple Customers At Same Location - When selected, if multiple customers are found at the same location in the source (drawing) file, they will be "stacked" in the model at the same location. When unselected, only one customer (the first customer encountered) will be allowed within the Fuzzy Tolerance of a given location.

Assign Text Value To Customer Link ID Number - When selected, If the *Customer Entity Type* is a "Text Location," the entity value will be assigned as the Link ID Number for the imported or merged customer features.

Customer Entity Type - Specifies the entity type used to represent the customer locations in the drawing. This item is only enabled if a value is selected for the *Customer Layer* item.

Customer Layer - Specifies whether customers will be processed during the Import or Merge routine. If customer locations are present in the drawing, select the layer's name containing the customer locations. If a layer is assigned, customers will be processed during the Import or Merge routine.

Drawing Coordinate Units - Specifies the dimensional units for the coordinate values in the source (drawing) file. It is used to make the appropriate conversion between pipe lengths, drawing coordinate units, and model Coordinate units.

Origin Shift - Specifies the amount to shift the coordinate values during the Import or Merge routine. The X-value indicates the horizontal (east-west) amount to shift the coordinates. The Y-value indicates the vertical (north-south) amount to shift the coordinates. The values are expressed in model Coordinate units.

Scale Factor - Specifies the value to scale the imported or merged coordinate values by. This item is automatically calculated if the *Drawing Coordinate Units* and *Model Coordinate Units* are specified.

Service Line Layer - Specifies the layer associated with lines representing service lines to be processed during the Import or Merge routine. If service lines are present in the drawing, select the layer's name containing the service locations.

Command Buttons

Cancel - Closes the screen without saving changes to the specification or proceeding with the associated routine.

Clear Data Field Assignments - Clears all of the Data Field Assignments. This command button is displayed on the *General Settings* data tab.

Close - Closes the screen, saves changes to the specification, and does not proceed with the associated routine.

Continue - Saves the current specification settings and proceeds with the associated routine.

Open Specification - Opens a previously saved specifications file (.dxi). When clicked, the File Selection screen is displayed.

Save Specification - Saves the current specification values. When clicked, the File Selection screen is displayed.

Set Default Values - Allows changes to the default data values. When clicked, the Default Data Values screen is displayed.

Notes & Considerations

● GASWorkS expects the source file to be in an AutoCAD-created drawing exchange format (DXF) file. However, files created by other DXF-compatible applications may also be acceptable.

● The DXF Import routine allows drawing exchange format files created by various CAD or other graphics packages to create a GASWorkS model. A complete model is created when the routine is used, including a Header, Customer (if included), Node, and Pipe Data file.

● GASWorkS constructs a new model by converting the linear graphic features on the "pipe" layers to pipes in the GASWorkS model. A pipe is created for each line, polyline, or arc feature on the specified "pipe" layers. The geographic/scaled length value of the original feature is used as the length for the new pipe-type feature. The specified pipe size is assigned to pipes processed from the associated layer.

● A node is placed at the beginning and end of each pipe (linear feature).

● The GASWorkS GDI supports coordinates from -1E+09 to 1E+09. If the DXF file contains coordinate values outside this range, they may be modified using the *Shift Model Coordinates* command after the file has been imported. However, it is best to correct the original drawing and recreate the DXF so that the modifications will not be repeated if the drawing is ever reimported.

● Do not leave empty rows between entries in the *Pipe Layers* list. Do not include extraneous data on the layers selected for importing. GASWorkS does not know the difference between a line intended to represent a pipe and a line intended to represent a street right-of-way. Or a block representing a customer location and a block representing a fire hydrant. Imported layers should contain only the graphics representing the features for which they have been selected.

● The Import routine cannot sense your intentions. The drawing used to create the DXF should accurately reflect the intended topology (connectivity) of the model to be created. The following table lists some common mistakes made when working with DXF files:

Problem	Comments
Lines are not broken at intersections in the drawing.	Nodes are only placed at the beginning and end of the linear entities found on the designated pipe layers. Often, a long, single, unbroken line is drawn along several street blocks. The laterals coming off this line will create nodes at their ends, and it will appear that there is a node at the intersection; however, because the line is connected to it, it is not broken at the intersections, and no connection is made at the lateral locations.

To remedy this problem, break all lines at all intersections where a connection to another line is intended. |
| Nodes appear at locations other than pipe intersections or pipe ends. | This often occurs because multi-segment lines are created using a combination of several line and/or arc segments. These types of lines should be constructed of polyline segments to accurately reflect the model connectivity.

To remedy this problem, convert and join the individual line segments into a single multi-segment polyline, which is continuous between the intended pipe ends. |
| Many nodes appear around regulator and regulator symbols. | This occurs because the symbol used to represent the regulator location is created using a small "drawing," the line work representing the symbol is contained on the designated pipe layer.

To remedy the problem, use "blocks" to depict the symbols and/or move the line work representing the symbols to a separate layer. |
| The length of pipes in the GASWorkS model is twelve times longer than they were in the original drawing. (DXF from AutoCAD) | This occurs when using AutoCAD and architectural units are used in the original drawing. When a DXF file is created from a drawing with these units, the resulting coordinate values are represented in inches instead of the standard "Feet" units.

To remedy this problem, change the drawing units to engineering and rescale the drawing. Or, set the appropriate *Drawing Coordinate Units* when importing the file. Or, rescale the model in GASWorkS after it has been imported - use a Scale Factor of 0.08333 (1/12). |

Problem	*Comments*
The coordinate values in the GASWorkS model are different from those in the original drawing. (DXF from MicroStation)	This occurs when creating a DXF file from MicroStation. MicroStation does not internally store the graphical representation of the drawing in true coordinates but uses positional units to represent the drawing. When the DXF is created, the positional units are reported in the file instead of the true coordinates (working units). To remedy this problem, set the Translation Units to Master when exporting (creating) the DXF from MicroStation. Or, appropriately rescale the model in GASWorkS after it has been imported.

● The GASWorkS GDI provides tools for repairing many of the problems listed in the previous table. However, it is best to correct the original drawing and recreate the DXF so that the modifications will not need to be repeated if the drawing is ever reimported.

● After importing, the *Flag Unbroken Intersections* command may be used to find and mark locations where the header pipe may not be broken (tapped) at a lateral intersection. Once flagged, the locations may be reviewed and manually corrected using the *Tap Pipe* command or automatically corrected using the *Break Flagged Intersections* command.

● Services can only be correctly imported from a DXF file containing pipes and customers. Services may be created from a DXF file if the existing model contains pipes and customers.

● When a value is assigned to the Valve Node Layer, block or point entities are processed during the Import or Merge routine. During the processing, the model is checked to see if a node exists at the entity's location; if it does, its Hydraulic Type is set to "Valve." If an existing node does not exist in the model, and the location falls within the Fuzzy Tolerance of an existing pipe in the model, the pipe is "tapped" at that entity's location, and its Hydraulic Type is set to "Valve."

Import GASWorkS 7.0, 8.0, 9.0 Files

GASWorkS 10 can read and import model files created by older versions of GASWorkS as far back as version 7.0. This may be done in two ways.

● From the *File* menu, select the *Open* item. The File Selection screen is displayed. If a Filename that is a GASWorkS 9.0 or older model is either selected or typed in the data field, a message will be displayed asking whether to import the file. Click the *Yes* command button to execute the appropriate Import routine for the GASWorkS version and create a set of model files with the same name as the original file followed by a "_10" suffix.

● From the Import screen, the GASWorkS 7.0, GASWorkS 8.0, or GASWorkS 9.0 *File Type* item may be selected. Once the *Import* command button is clicked, the File Selection screen is displayed. Select the name of the desired model. If the selected model is a different version than was specified (for example, the GASWorkS 9.0 Import routine was selected, but a GASWorkS 8.0 file was selected), a message will be displayed, and the routine will not be executed.

GASWorkS 10 will create a new version of the model using the data from the older model. The older model files will be copied or updated and placed in the directory (folder) of the new model under the new model name.

Due to differences between the versions, some settings may not be carried over from the older GASWorkS model when imported into GASWorkS 10. Also, new features in GASWorkS 10 that were unavailable in older versions may be set to default values.

Import Or Merge KML (Keyhole Markup Language) Files

When importing or merging a KML (Keyhole Markup Language) file, certain information must be specified about the selected file. The import or merge specifications and settings for a KML file are defined using the Import Or Merge Specifications screen. The screen is displayed when the *KML (Keyhole Markup Language) File* File Type item is selected from the Import screen or the Merge screen. The features associated with the screen are described as follows.

Import Or Merge Specifications Screen - *Pipe Settings*

Data Items

Fuzzy Tolerance - Specifies the *Fuzzy Tolerance* value when processing the source file. Line ends that fall within the *Fuzzy Tolerance* value are snapped together. The value is expressed in the geographic coordinate units (decimal degrees).

Pipe Folders - Displays the Folder Assignment and Pipe Size assignments. To make an assignment - under the *Folder Assignment* column, click on the associated cell and select an item from the list. Click in the adjacent cell under the *Pipe Size* column and select an item from the list to complete the assignment.

Folder Assignment - Displays the selected folders from the KML file.

Pipe Size - Displays the selected GASWorkS data items.

Pipe Length Units - Specifies the dimensional units for the length values of the imported or merged pipes. It is used to make the appropriate conversion between pipe lengths, drawing coordinate units, and model units.

Import Or Merge Specifications Screen - *Other Settings*

Data Items

Allow Multiple Customers At Same Location - When selected, if multiple customers are found at the same location in the source file, they will be "stacked" in the model at the same location. When unselected, only one customer (the first customer encountered) will be allowed within the Fuzzy Tolerance of a given location. This item is only enabled if a value is selected for the *Customer Folder* item.

Customer Folder - Specifies whether customers will be processed during the Import or Merge routine. If customer locations are present in the file, select the name of the folder containing the customers. If a folder is assigned, customers will be processed during the Import or Merge routine.

Command Buttons

Cancel - Closes the screen without saving changes to the specification or proceeding with the associated routine.

Clear Data Field Assignments - Clears all of the Data Field Assignments. This command button is displayed on the *General Settings* data tab.

Close - Closes the screen, saves changes to the specification, and does not proceed with the associated routine.

Continue - Saves the current specification settings and proceeds with the associated routine.

Open Specification - Opens a previously saved specifications file (.sxi). When clicked, the File Selection screen is displayed.

Save Specification - Saves the current specification values. When clicked, the File Selection screen is displayed.

Set Default Values - Allows changes to the default data values. When clicked, the Default Data Values screen is displayed.

Notes & Considerations

● Items in the folders selected in the *Folder Assignment* cells will be imported and assigned the pipe sizes indicated in the adjacent *Pipe Size* cells. Items in a folder not assigned to a pipe size will not be imported.

● GASWorkS expects the source file to be a Keyhole Markup Language (KML) file, such as those used by Google Earth. Any file that follows the official KML schema may be used.

● GASWorkS expects pipes and customers to be saved in "folders" within the file.

● GASWorkS constructs the new model by turning the LineString features in the assigned "Pipe" folders into pipes in the GASWorkS model. All pipes are imported as polylines. The geographic/scaled length value of the original feature is used as the length for the new pipe feature. The specified pipe size is assigned to pipes processed from the associated folder.

● A node is placed at the beginning and end of each pipe (linear feature).

● Customers are imported by turning the point features found in the assigned "Customer" folder into customer locations in the GASWorkS model. If a customer folder is assigned, a message will be displayed asking if the supply main connections are to be assigned during the Import routine. Click the *Yes* command button to have the service line connections assigned or the *No* command button to leave the customers unassigned.

● Do not leave empty rows between entries in the *Pipe Folders* table. Do not include extraneous data in the folders selected for importing. GASWorkS does not know the difference between a line intended to represent a pipe and a line intended to represent a street right-of-way. Or a block representing a customer location and a block representing a fire hydrant. Imported folders should contain only the graphics representing the features for which they have been selected.

● Features drawn in Google Earth do not automatically snap to each other, and lines that appear connected may not be connected upon close inspection. GASWorkS cannot automatically correct all of these issues. Models imported from KML files may experience connectivity issues. Use the tools in GASWorkS to find and connect connectivity issues.

● See the *Notes & Considerations* section of the DXF Import Routine for a list of common errors and issues that might be found in imported data.

GASWorkS™ 10.1 *Import, Merge, & Purge*

Import Or Merge SHP (Shapefiles) Files

GASWorkS provides a routine for importing or merging GIS files, meeting the shapefile (SHP) format of the Environmental Systems Research Institute (ESRI). The files may be created by an ESRI GIS application or by another application that supports the SHP file format.

Customer Import Or Merge Specifications Screen - *General Settings*

When importing or merging a customer shapefile (SHP) file, certain information about the selected file must be specified. The specifications and settings for importing or merging a customer SHP file are defined using the associated Import or Merge Specifications screen. The screens are displayed when a *SHP (ESRI Shapefile) File - Customer* type item is selected during the Import or Merge routines. The features associated with the screens are described as follows.

GASWorkS™ 10.1   *Import, Merge, & Purge*

Displayed Items

Model Coordinate Units - Displays the dimensional units for the coordinate values in the model file. It is used to make the appropriate conversion between lengths, source coordinate units, and model Coordinate units. Click the *Set Default Values* command button to change the model Coordinate units.

Data Items

Data Field Assignments - Displays the association between fields in the SHP file and data items in GASWorkS. To make an assignment - in the *Database Field Assignment* column, click in a cell and select an item from the list; in the adjacent cell, in the *GASWorkS Data Item* column, select an item from the list.

> **Database Field Assignment -** Displays and allows the selection of fields from the SHP file.

> **GASWorkS Data Item -** Displays and allows the selection of GASWorkS data items.

Origin Shift - Specifies the amount to shift the source coordinate values during the Import or Merge routine. The X-value indicates the horizontal (east-west) amount to shift the coordinates. The Y-value indicates the vertical (north-south) amount to shift the coordinates. The values are expressed in model Coordinate units.

Scale Factor - Specifies the value to scale the source coordinate values by.

Source File Coordinate Units - Specifies the dimensional units for the coordinate values in the source file. This item is used to make the appropriate conversion between coordinate values contained in the source file and the model.

Customer Import Or Merge Specifications Screen - *Other Settings*

Data Items

Assign Service Lines During Import/Merge - When selected, the "service lines" will be created using the selected *Assignment Method* during the Import or Merge routine.

Assignment Method - Specifies the supply main selection method when service lines are assigned during the Import or Merge routine. See the *Notes & Considerations* section for a description of the assignment methods.

Fuzzy Tolerance - Specifies the *Fuzzy Tolerance* value when processing the source file. Only displayed using the Merge routine and when the "By Coordinate Location" option is selected for the *Search Method*.

Load Assignment Method When Match Found - Specifies the method to be used when a matching customer location is found during the Merge routine. See the *Notes & Considerations* section for a description of the assignment methods.

Maximum Service Line Length - Specifies the maximum search distance using the "Nearest Main" assignment method. The value is expressed in model Coordinate units.

Multiply Load Value By (-1) During Import/Merge - When selected, if a load value is assigned, the value will be multiplied by -1 during the Import or Merge routine.

Search Method - Specifies the method used to search for a "matching" (existing) customer during the Merge routine. See the *Notes & Considerations* section for a description of the search methods.

Command Buttons

Cancel - Closes the screen without saving changes to the specification or proceeding with the associated routine.

Clear Data Field Assignments - Clears all of the Data Field Assignments. This command button is displayed on the *General Settings* data tab.

Close - Closes the screen, saves changes to the specification, and does not proceed with the associated routine.

Continue - Saves the current specification settings and proceeds with the associated routine.

Open Specification - Opens a previously saved specifications file (.sxi). When clicked, the File Selection screen is displayed.

Save Specification - Saves the current specification values. When clicked, the File Selection screen is displayed.

Set Default Values - Allows changes to the default data values. When clicked, the Default Data Values screen is displayed.
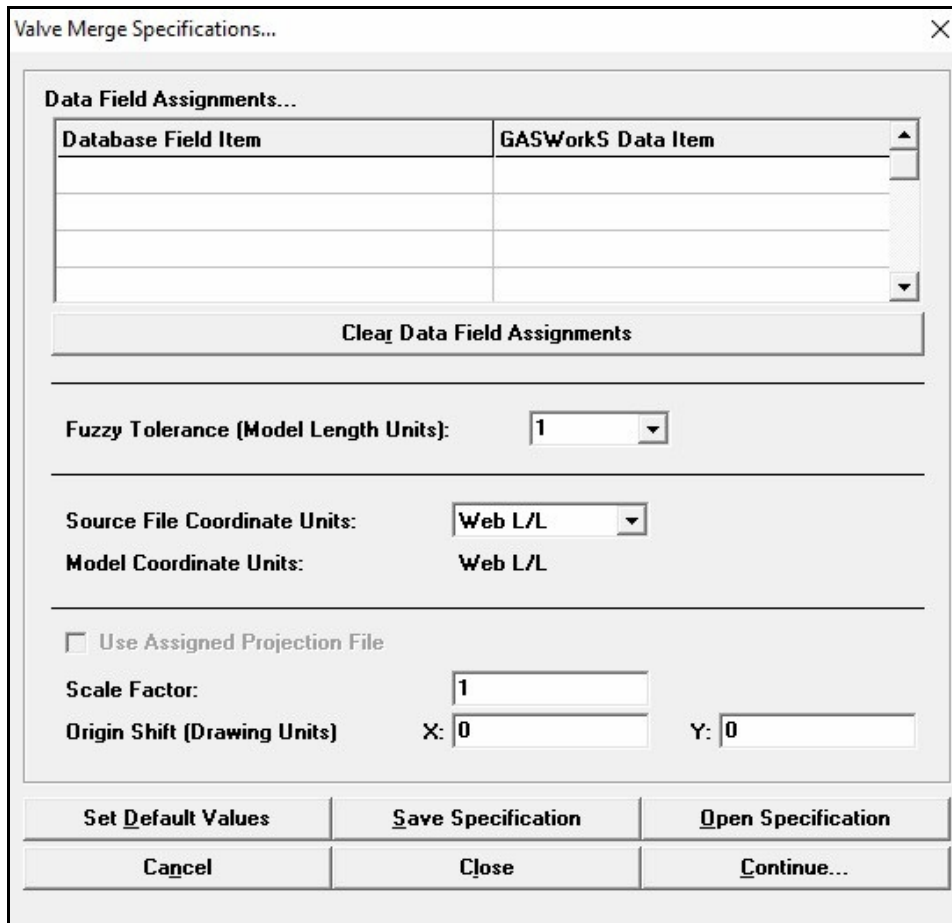

Notes & Considerations

● Do not leave empty rows between entries in the *Data Field Assignments* list.

● When merging a customer data file into an existing model, search options are provided to determine if a customer in the data file already exists in the model. If it does, the *Load Per Unit* value of the matching customer is updated according to the specified *Load Assignment Method When Match Found*. If a match is not found, a new customer feature is not created in the model, and its data values are assigned accordingly. The search options and their behavior are listed below.

> *None - Do Not Search For Match* - When selected, no search is performed. When this option is used, a customer will be placed regardless of whether a customer already exists at the same location.

> *By Coordinate Location* - When selected, a search is performed to determine if the customer location from the data file falls within the specified Fuzzy Tolerance of an existing customer in the model. If it does, it is considered to be a "match."

> *By Link ID Number* - When selected, a search is performed to determine if the Link ID Number associated with the customer from the data file is already assigned to an existing customer in the model. If it does, it is considered to be a "match."

● When merging a customer data file into an existing model, using the "By Coordinate Location" or "By Link ID Number" search options, when a match is found, load values from the data file are assigned according to the selected "Load Assignment Method When Match Found. The assignment methods and their behavior are listed below.

> *Add To Existing Value* - When selected, when a match is found, the value from the data field assigned to the *Load Per Unit* GASWorkS Data Item will be added to the current *Per Unit Load* value for the matching customer in the model.

> *Do Not Change Existing Value* - When selected, when a match is found, no action is taken. The *Per Unit Load* value for the matching customer in the model is unchanged.

> *Replace Existing Value* - When selected, when a match is found, the current *Per Unit Load* value for the matching customer in the model will be replaced with the value from the data field assigned to the *Load Per Unit* GASWorkS Data Item.

> *Use Larger Value* - When selected, when a match is found, the current *Per Unit Load* value for the matching customer in the model will be compared to the value from the data field assigned to the *Load Per Unit* GASWorkS Data Item. The current *Per Unit Load* value for the matching customer in the model will be set to the larger of the two values.

> *Note* - If no data field is assigned to the *Load Per Unit* GASWorkS Data Item, the current *Per Unit Load* value for the matching customer in the model will not be changed.

● When coordinate values are assigned in the *Data Field Assignments* list, the values read from the data file will be changed as follows:

> • The coordinate values will first be multiplied by the specified Scale Factor value.

> • The scaled coordinate values will then be increased or decreased by the values specified for the Origin Shift values.

● The customer data is processed using the Import and Merge routines as follows:

> • The customer locations are taken from the geographic data file (SHP file). The file must contain the point or polygon-type data features. After the customer locations have been created, the attribute data is read from the attribute data file (.dbf).

> • When the Merge routine is chosen, matching between the existing model and the new data is performed geographically. If an existing customer is found within the Fuzzy Tolerance of a new customer's X-Y location, a match is assumed.

● Generally, the customer data should be merged into the model after the pipe data has been created and cleaned up.

Node Merge Specifications Screen - *Other Settings*

Data Items

Fuzzy Tolerance - Specifies the *Fuzzy Tolerance* value when processing the source file. Only displayed when the "By Coordinate Location" option is selected for the *Search Method*.

Load Assignment Method When Match Found - Specifies the method to be used when a matching node location is found during the Merge routine. See the *Notes & Considerations* section for a description of the assignment methods.

Multiply Load Value By (-1) During Merge - When selected, if a load value is assigned, the value will be multiplied by -1 during the Merge routine.

Search Method - Specifies the method used to search for a "matching" (existing) node during the Merge routine. See the *Notes & Considerations* section for a description of the search methods.

Command Buttons

Cancel - Closes the screen without saving changes to the specification or proceeding with the associated routine.

Clear Data Field Assignments - Clears all of the Data Field Assignments. This command button is displayed on the *General Settings* data tab.

Close - Closes the screen, saves changes to the specification, and does not proceed with the associated routine.

Continue - Saves the current specification settings and proceeds with the associated routine.

Open Specification - Opens a previously saved specifications file (.sxi). When clicked, the File Selection screen is displayed.

Save Specification - Saves the current specification values. When clicked, the File Selection screen is displayed.

Set Default Values - Allows changes to the default data values. When clicked, the Default Data Values screen is displayed.

Notes & Considerations

● The most common use of the Node Merge routine is to update load values in an existing model when certain nodes represent customer demand locations or to update input flow rates when certain nodes represent supply points like production wells.

● Do not leave empty rows between entries in the *Data Field Assignments* list.

● When merging a node data file into an existing model, search options are provided to determine if a node in the data file already exists in the model. If it does, the *Base Load* value of the matching node is updated according to the specified *Load Assignment Method When Match Found*. The search options and their behavior are listed below.

> *By Coordinate Location* - When selected, a search is performed to determine if the node location from the data file falls within the specified Fuzzy Tolerance of an existing node in the model. If it does, it is considered to be a "match."

> *By Node Name* - When selected, a search is performed to determine if the *Name* value associated with the node from the data file is already assigned to an existing node in the model. If it does, it is considered to be a "match."

● When merging a node data file into an existing model using the "By Coordinate Location" or "By Node Name" search options, when a match is found, load values from the data file are assigned according to the selected *Load Assignment Method When Match Found* method. The assignment methods and their behavior are listed below.

> *Add To Existing Value* - When selected, when a match is found, the value from the data field assigned to the *Load, Base* GASWorkS Data Item, will be added to the current *Base Load* value for the matching node in the model.

> *Replace Existing Value* - When selected, when a match is found, the current *Base Load* value for the matching node in the model will be replaced with the value from the data field assigned to the *Load, Base* GASWorkS Data Item.

> *Use Larger Value* - When selected, when a match is found, the current *Base Load* value for the matching node in the model will be compared to the value from the data field assigned to the *Load, Base* GASWorkS Data Item. The current *Base Load* value for the matching node in the model will be set to the larger of the two values.

> *Note* - If no data field is assigned to the *Load, Base* GASWorkS Data Item, the current *Base Load* value for the matching node in the model will not be changed.

● Coordinate values read from the data file will be changed as follows:

> • The coordinate values will first be multiplied by the specified Scale Factor value.

> • The scaled coordinate values will then be increased or decreased by the values specified for the Origin Shift values.

Pipe Import Or Merge Specifications Screen

When importing or merging a pipe shapefile (SHP) file, certain information about the selected file must be specified. The specifications and settings for importing or merging a pipe SHP file are defined using the associated Pipe Import or Merge Specifications screens. The screens are displayed when a *SHP (ESRI Shapefile) File - Pipe* type item is selected during the Import and Merge routines. The features associated with the screens are described as follows.

Displayed Items

Model Coordinate Units - Displays the dimensional units for the coordinate values in the model file. It is used to make the appropriate conversion between lengths, source coordinate units, and model Coordinate units. Click the *Set Default Values* command button to change the model Coordinate units.

GASWorkS™ 10.1 *Import, Merge, & Purge*

Data Items

Data Field Assignments - Displays the association between fields in the SHP file and data items in GASWorkS. To make an assignment - in the *Database Field Item* column, click in a cell and select an item from the list; in the adjacent cell, in the *GASWorkS Data Item* column, select an item from the list.

> **Database Field Item -** Displays and allows the selection of fields from the SHP file.

> **GASWorkS Data Item -** Displays and allows the selection of GASWorkS data items.

Fuzzy Tolerance - Specifies the *Fuzzy Tolerance* value when processing the source file.

Origin Shift - Specifies the amount to shift the source coordinate values during the Import or Merge routine. The X-value indicates the horizontal (east-west) amount to shift the coordinates. The Y-value indicates the vertical (north-south) amount to shift the coordinates. The values are expressed in model Coordinate units.

Scale Factor - Specifies the value to scale the source coordinate values by.

Source File Coordinate Units - Specifies the dimensional units for the coordinate values in the source file. This item is used to make the appropriate conversion between coordinate values contained in the source file and the model.

Use Assigned Projection File - Indicates whether the assigned projection file will be used during the Import or Merge routine.

Command Buttons

Cancel - Closes the screen without saving changes to the specification or proceeding with the associated routine.

Clear Data Field Assignments - Clears all of the Data Field Assignments. This command button is displayed on the *General Settings* data tab.

Close - Closes the screen, saves changes to the specification, and does not proceed with the associated routine.

Continue - Saves the current specification settings and proceeds with the associated routine.

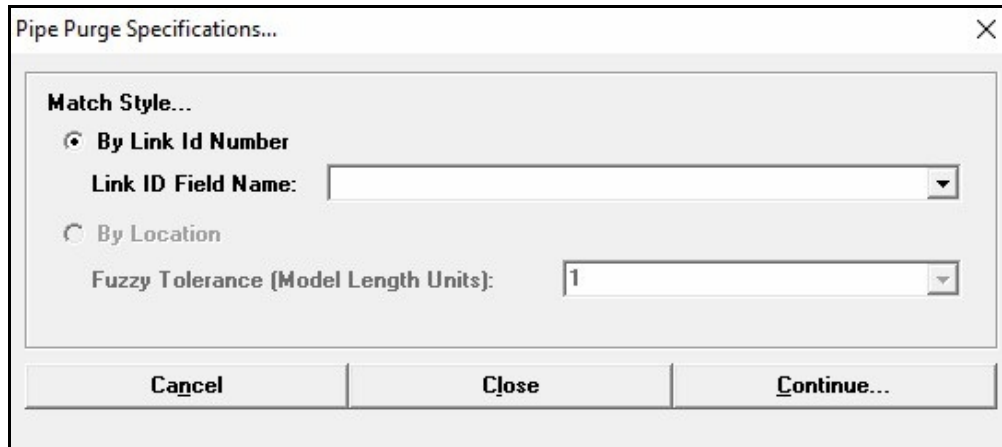**Open Specification -** Opens a previously saved specifications file (.sxi). When clicked, the File Selection screen is displayed.

Save Specification - Saves the current specification values. When clicked, the File Selection screen is displayed.

Set Default Values - Allows changes to be made to the default data values. When clicked, the Default Data Values screen is displayed.

Notes & Considerations

● An advantage of using the shapefile (SHP) file format is that it contains graphic and attribute information. A shapefile consists of at least three separate files:

> • The geographic data (shapefile), that ends in a ".shp" extension - contains the geographic information.

> • The data file that ends in a ".dbf" extension - contains the attribute data in a dBase III format.

> • The index file that ends in an ".shx" extension - contains relationship information that links the geographic and attribute data.

> • Optionally, the projection file, that ends in an ".prj" extension - contains information about the coordinate system used in the geographic data file.

> • Other files may also exist.

● GASWorkS supports shapefiles containing pipe, junction, customer, service, regulator, and valve data. A basic GASWorkS model may be created from just a pipe file. If other data is available, it may be merged to create a more comprehensive model.

● When creating a model from a set of shapefiles, the general order for processing the various files is - pipe data, junction data (if used), regulator data, customer data, service data, and valve data. At a minimum, a pipe file must be included. Regardless of which files are used, the pipe file should be imported first, and the subsequent files should be merged into the resulting model. It is suggested that the model be reviewed after the pipe data has been processed. Any configuration or connectivity issues should be resolved before processing additional data types.

● If there is a projection file associated with the shapefile, GASWorkS will copy the existing projection file and create a new one associated with the model. Please note that this only occurs when a pipe shapefile is imported.

● Do not leave empty rows between entries in the *Data Field Assignments* list.

● The pipe data is processed using the Import and Merge routines as follows:

> • The pipe locations are taken from the geographic data file (SHP file). The file must contain arc-type data features. After the pipe locations have been created, the attribute data is read from the attribute data file (.dbf).

• When the Merge routine is selected, matching between the existing model and the new data is performed geographically. If an existing pipe's From Node and To Node, and all of its vertices, are found within the Fuzzy Tolerance of a new pipe's node X-Y locations, a match is assumed.

• Generally, the pipe data should be created before any other data is merged into the model.

● Coordinate values read from the data file will be changed as follows:

• The coordinate values will first be multiplied by the specified Scale Factor value.

• The scaled coordinate values will then be increased or decreased by the values specified for the Origin Shift values.

● If the GIS database that the pipe shapefile was created from is using the "geometric network" method, the associated "junction" file should be merged into the model to finish creating the pipe connectivity. When merging in junction data, pipe intersections will be "broken" where a junction is found at an "untapped" lateral pipe location.

● See the *Notes & Considerations* section of the DXF Import Routine for a list of common errors and issues that might be found in the imported data.

GASWorkS™ 10.1   *Import, Merge, & Purge*

Service Merge Specifications Screen

When merging a service line shapefile (SHP) file, certain information must be specified about the selected file. The specifications and settings for merging a service line SHP file are defined using the Service Merge Specifications screen. The screen is displayed when the *SHP (ESRI Shapefile) File - Service* item is selected on the Merge screen. The features associated with the screen are described as follows.

Displayed Items

Model Coordinate Units - Displays the dimensional units for the coordinate values in the model file. It is used to make the appropriate conversion between lengths, source coordinate units, and model Coordinate units. Click the *Set Default Values* command button to change the model Coordinate units.

Data Items

Data Field Assignments - Not used. Disabled for this file type.

Fuzzy Tolerance - Specifies the *Fuzzy Tolerance* value to use when processing the source file.

Origin Shift - Specifies the amount to shift the source coordinate values during the Merge routine. The X-value indicates the horizontal (east-west) amount to shift the coordinates. The Y-value indicates the vertical (north-south) amount to shift the coordinates. The values are expressed in model Coordinate units.

Scale Factor - Specifies the value to scale the source coordinate values.

Source File Coordinate Units - Specifies the dimensional units for the coordinate values in the source file. This item is used to make the appropriate conversion between coordinate values contained in the source file and the model.

Use Assigned Projection File - Not used. Disabled for this file type.

Command Buttons

Cancel - Closes the screen without saving changes to the specification or proceeding with the associated routine.

Clear Data Field Assignments - Clears all of the Data Field Assignments. This command button is displayed on the *General Settings* data tab.

Close - Closes the screen, saves changes to the specification, and does not proceed with the associated routine.

Continue - Saves the current specification settings and proceeds with the associated routine.

Open Specification - Opens a previously saved specifications file (.sxi). When clicked, the File Selection screen is displayed.

Save Specification - Saves the current specification values. When clicked, the File Selection screen is displayed.

Set Default Values - Allows changes to the default data values. When clicked, the Default Data Values screen is displayed.

GASWorkS™ 10.1   *Import, Merge, & Purge*

Notes & Considerations

● To effectively use a service line shapefile, the general order for processing the various files is - pipe data, junction data, customer data, and service line data.

● Arcs representing the service lines should be continuous from the tap location to the customer location.

● The Merge routine is performed as follows:

     • The geographic data for each "arc" in the shapefile is read.

     • The beginning end of the line is examined to see if a pipe or customer location falls within the specified Fuzzy Tolerance of the end.

         • If a pipe feature is found, that end is considered the "tap" location; the other end is examined to see if a customer feature falls within the Fuzzy Tolerance of that end. If it does, the service line is added to the model and associated with the matching customer.

         • If a customer is found, that end is considered the "customer" location; the other end is examined to see if a pipe feature falls within the Fuzzy Tolerance of that end. If it does, the service line is added to the model and associated with the matching customer.

● Coordinate values read from the data file will be changed as follows:

     • The coordinate values will first be multiplied by the specified Scale Factor value.

     • The scaled coordinate values will then be increased or decreased by the values specified for the Origin Shift values.

GASWorkS™ 10.1 — *Import, Merge, & Purge*

Valve Merge Specifications Screen

When merging a shapefile (SHP) representing valve locations, certain information must be specified about the selected file. The specifications and settings for merging a valve SHP file are defined using the Valve Merge Specifications screen. The screen is displayed when the *SHP (ESRI Shapefile) File - Valve* item is selected on the Merge screen. The features associated with the screen are described as follows.

Displayed Items

Model Coordinate Units - Displays the dimensional units for the coordinate values in the model file. It is used to make the appropriate conversion between lengths, source coordinate units, and model Coordinate units. Click the *Set Default Values* command button to change the model Coordinate units.

GASWorkS™ 10.1 — *Import, Merge, & Purge*

Data Items

Data Field Assignments - Displays the association between fields in the SHP file and data items in GASWorkS. To make an assignment - in the *Database Field Item* column, click in a cell and select an item from the list; in the adjacent cell, in the *GASWorkS Data Item* column, select an item from the list.

> **Database Field Item -** Displays and allows the selection of fields from the SHP file.

> **GASWorkS Data Item -** Displays and allows the selection of GASWorkS data items.

Fuzzy Tolerance - Specifies the *Fuzzy Tolerance* value when processing the source file.

Origin Shift - Specifies the amount to shift the source coordinate values during the Merge routine. The X-value indicates the horizontal (east-west) amount to shift the coordinates. The Y-value indicates the vertical (north-south) amount to shift the coordinates. The values are expressed in model Coordinate units.

Scale Factor - Specifies the value to scale the source coordinate values.

Source File Coordinate Units - Specifies the dimensional units for the coordinate values in the source file. This item is used to make the appropriate conversion between coordinate values contained in the source file and the model.

Use Assigned Projection File - Not used. Disabled for this file type.

Command Buttons

Cancel - Closes the screen without saving changes to the specification or proceeding with the associated routine.

Clear Data Field Assignments - Clears all of the Data Field Assignments. This command button is displayed on the *General Settings* data tab.

Close - Closes the screen, saves changes to the specification, and does not proceed with the associated routine.

Continue - Saves the current specification settings and proceeds with the associated routine.

Open Specification - Opens a previously saved specifications file (.sxi). When clicked, the File Selection screen is displayed.

Save Specification - Saves the current specification values. When clicked, the File Selection screen is displayed.

Set Default Values - Allows changes to be made to the default data values. When clicked, the Default Data Values screen is displayed.

Notes & Considerations

● When using the valve shapefile (SHP) file merge routine, valve locations must be represented as point features in the shapefile. The Valve Merge routine should not be executed until the pipe features have been created and "cleaned up."

● The Merge routine is performed as follows:

• The geographic data for each "point" in the shapefile is read.

• The point's location is examined to see if an existing pipe or node feature falls within the specified Fuzzy Tolerance of its location.

• If a pipe feature is found, the pipe is "tapped" at the point location, and a new node is added with its Hydraulic Type set to "Valve."

• If a node feature is found, the node's Hydraulic Type is set to "Valve."

• If no feature is found, the point will be ignored.

• When a new node is added or an existing node is found, the value assigned to a data file for the "Valve/Node Number" is assigned to the associated node.

● Coordinate values read from the data file will be changed as follows:

• The coordinate values will first be multiplied by the specified Scale Factor value.

• The scaled coordinate values will then be increased or decreased by the values specified for the Origin Shift values.

GASWorkS™ 10.1 *Import, Merge, & Purge*

Import Stoner PD/XY Files

GASWorkS provides a routine for importing Stoner & Associates "PD" and "XY" format files. The files may be created by a Stoner modeling package or another package supporting the PD file format. The routine is based on our best knowledge of the Stoner files.

There are some variations in the PD file format, depending on the version of the application that created the file. Some files may or may not import correctly.

Not all options supported by the Stoner modeling software are supported by GASWorkS; messages may appear during the Import routine warning of unsupported options. After the import, you must correct any inconsistencies resulting from unsupported options before attempting to solve the imported model.

If an XY file is detected during the Import routine, the graphical data in the XY file will be assigned to the pipes and nodes imported from the PD file.

Purge Routine

This section provides descriptions and requirements for purging features from a model using data from a supported data file. The additional data screens used to further define the purge specifications will be described later in this section.

The Purge routine reads data from the specified data file and attempts to find a matching feature in the current model. If a match is found, the feature is deleted from the model. The data file should only contain records for the features to be removed.

The following table describes the Purge *File Types* supported by GASWorkS.

File Type
SHP (ESRI Shapefile) File - Customer, Pipe

The Purge routine is accessed through the *Purge* item of the *Utilities* menu. When selected, the Purge screen is displayed. The features associated with the Purge screen, along with the specific requirements associated with each supported file type, are described in the following sections.

GASWorkS™ 10.1

Purge Screen

Displayed Items

Purge Log - Displays the activity (status) of the Purge routine.

Processing Record - Displays the number of the record currently being processed or the number of the records that were processed when completed.

Data Items

File Type - Displays the type of file to be merged.

Command Buttons

Close - Closes the screen.
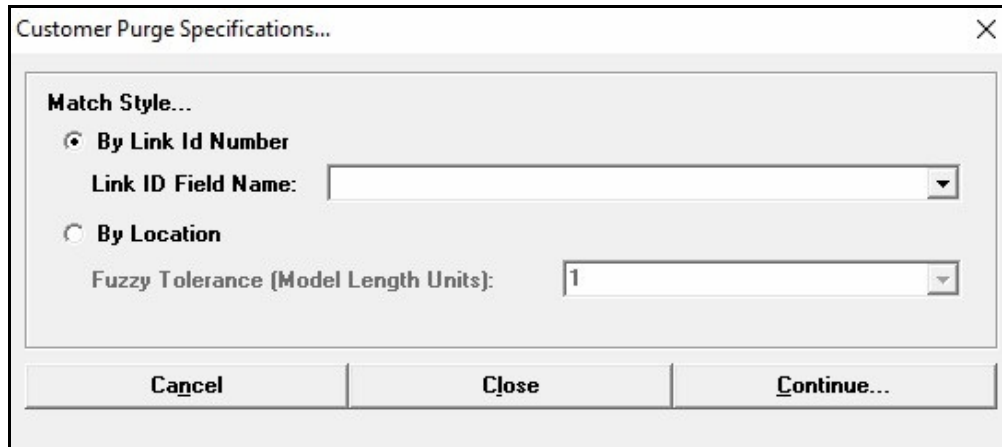
Purge - Starts the Purge routine. When selected, displays the File Selection screen.

Notes & Considerations

● The Purge routine allows features to be removed from the currently open model based on identifying data contained in a data file. When using the routine, the purge file is compared to the model. Where matching records are found, the matching feature in the model is deleted. Deleted features may be restored using the *Undo* command.

● Before processing the specified data file, an additional specification screen is displayed. Enter the appropriate data and settings to continue.

Customer Purge Specifications Screen

Data Items

Fuzzy Tolerance - Specifies the *Fuzzy Tolerance* value when processing the source file. Only displayed when the "By Location" option is selected for the *Match Style*.

Link ID Field Name - Specifies the name of the data field corresponding to the Link ID value in the selected data file. Only displayed when the "By Link ID Number" option is selected for the *Match Style*.

Match Style - Specifies the method used to search for a "matching" (finding) an existing customer during the Purge routine. See the following *Notes & Considerations* section for a description of the search methods.

Command Buttons

Cancel - Closes the screen without saving changes to the specification or proceeding with the associated routine.

Close - Closes the screen, saves changes to the specification, and does not proceed with the associated routine.

Continue - Saves the current specification settings and proceeds with the associated routine.

Notes & Considerations

● When purging using a customer data file, search options are provided to determine if a customer in the data file exists in the model. If it does, the feature will be deleted from the model. If a match is not found, no action is taken. The search options and their behavior are listed below:

By Link ID Number - When selected, a search is performed to determine if the Link ID Number associated with the customer from the data file is already assigned to an existing customer in the model. If it does, it is considered to be a "match."

By Location - When selected, a search is performed to determine if the customer location from the data file falls within the specified Fuzzy Tolerance of an existing customer in the model. If it does, it is considered to be a "match."

● The Purge routine works well when the model data is created from the same GIS data that the purge data file was created from. If locations or identification numbers have been changed either in the model or GIS data since the model was built, the Purge routine may not be effective.

Pipe Purge Specifications Screen

Data Items

Fuzzy Tolerance - Specifies the *Fuzzy Tolerance* value when processing the source file. Only displayed when the "By Location option" is selected for the *Match Style*.

Link ID Field Name - Specifies the name of the data field corresponding to the Link ID value in the selected data file. Only displayed when the "By Link ID Number" option is selected for the *Match Style*.

Match Style - Specifies the method used to search for a "matching" (finding) and existing pipe during the Purge routine. See the following *Notes & Considerations* section for a description of the search methods.

Command Buttons

Cancel - Closes the screen without saving changes to the specification or proceeding with the associated routine.

Close - Closes the screen, saves changes to the specification, and does not proceed with the associated routine.

Continue - Saves the current specification settings and proceeds with the associated routine.

Notes & Considerations

● When purging using a pipe data file, search options are provided to determine if a pipe in the data file exists in the model. If it does, the feature will be deleted from the model. If a match is not found, no action is taken. The search options and their behavior are listed below.

By Link ID Number - When selected, a search is performed to determine if the Link ID Number associated with the pipe from the data file is already assigned to an existing pipe in the model. If it does, it is considered to be a "match."

By Location - When selected, a search is performed to determine if the pipe location from the data file falls within the specified Fuzzy Tolerance of an existing pipe in the model. If it does, it is considered to be a "match." For a match to be found, the pipe end points and all vertices must fall within the Fuzzy Tolerance of each other.

● The Purge routine works well when the model data is created from the same GIS data that the purge data file was created from. If locations or identification numbers have been changed either in the model or GIS data since the model was built, the Purge routine may not be effective.

EXPORT ROUTINES

GASWorkS™ 10.1

Export Routine

This section provides descriptions and requirements for exporting data from GASWorkS into a supported file format. The additional data screens used to further define the export specifications will be described later in this section. The Help Guide contains various step-by-step examples of exporting the various data files.

GASWorkS provides translation routines for exporting data associated with several file formats. The following table describes the Export *File Types* supported by GASWorkS.

File Type	Supported By	
	Export	*Quick Export*
CSV (Comma Separated Values) File - Customer	✔	
CSV (Comma Separated Values) File - Node	✔	
CSV (Comma Separated Values) File - Pipe	✔	
CSV File Set		✔
DBF (dBase Format) File - Customer	✔	
DBF (dBase Format) File - Node	✔	
DBF (dBase Format) File - Pipe	✔	
DBF 3.0 File Set		✔
DXF (Drawing Exchange Format) File	✔	
GASWorkS 9.0 Model	✔	✔
KML (Keyhole Markup Language) File	✔	
SHP (ESRI Shapefile) File - Customer	✔	
SHP (ESRI Shapefile) File - Node	✔	
SHP (ESRI Shapefile) File - Pipe	✔	
SHP File Set		✔
Stoner PF/XY File	✔	
XLS 8.0 Workbook		✔
XLS (Excel) 8.0 & 97 Workbook - Customer, Node, Pipe	✔	

File Type	Supported By	
	Export	*Quick Export*
XLS (Excel) 8.0 & 97 Workbook - Customer	✔	
XLS (Excel) 8.0 & 97 Workbook - Node	✔	
XLS (Excel) 8.0 & 97 Workbook - Pipe	✔	

There are two methods to initiate the Export routine - either by using the Quick Export routine in the *File* menu or the Export routine in the *Utilities* menu.

The Quick Export routines are accessed using the *Quick Export* submenu of the *File* menu. The Quick Export routines automatically export the customer, node, and pipe data contained in the model to a single file or set of files in the specified format, depending on the target data File Type. When using the Quick Export routines, the data files associated with the currently open model are used to create the export files.

The Export routine is accessed by selecting the *Export* item in the *Utilities* menu. When selected, the Export screen is displayed. The features associated with the Export screen, along with the specific requirements associated with each supported file type, are described in the following sections.

GASWorkS™ 10.1 *Export Routines*

Export Screen
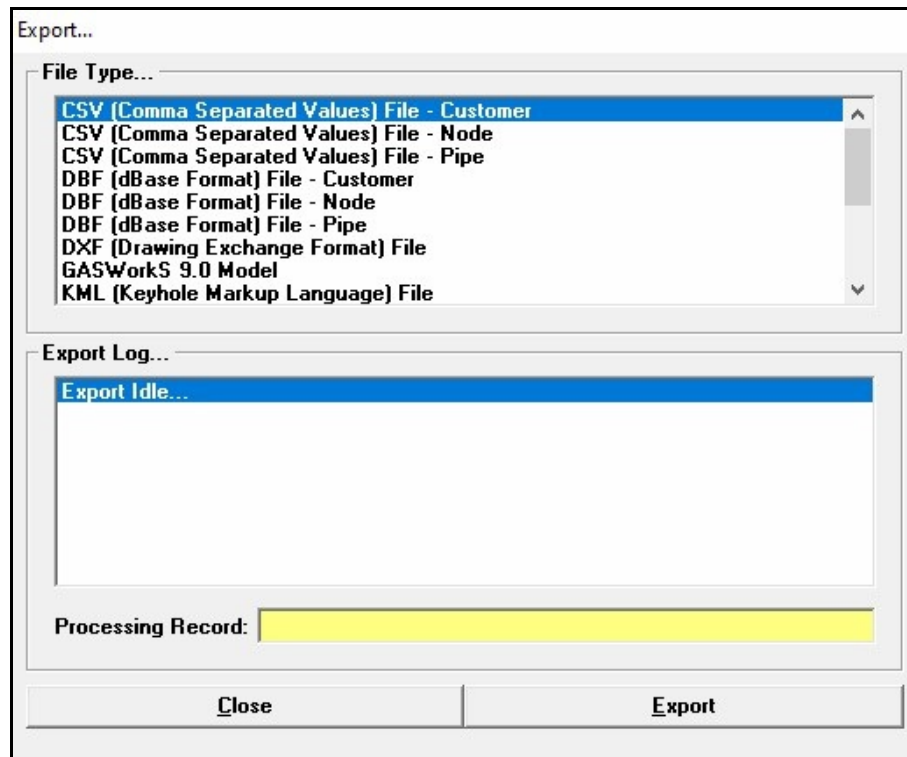
Displayed Items

Export Log - Displays the activity (status) of the Export routine.

Processing Record - Displays the number of the record currently being processed or the number of the records that were processed when completed.

Data Items

File Type - Displays the available file types that may be exported.

Command Buttons

Close - Closes the screen.

Export - Begins the Export routine. When clicked, the File Selection screen is displayed.

Notes & Considerations

● The Export routine creates a new set of files in the specified format; it does not change the format or content of the GASWorkS model data files.

Export CSV (Comma-Separated Values) Files

Pipe, node, and customer data may be exported to a CSV (Comma-Separated Values) file. Each feature type is exported to a separate file. No specifications are required to export to a CSV file.

When exporting to a CSV file using the Export routine, an individual file will be created for the selected model feature. When exported using the Quick Export routine, a file set is created - one file for each associated data feature. The resulting files may be opened and manipulated using any application which supports the CSV format.

Export DBF (dBase Format) Files

GASWorkS provides a routine for creating dBase III (DBF) files from a GASWorkS model. The routine is available using either the Export item in the Utilities menu or the appropriate item from the Quick Export submenu of the File menu. This routine is automatically executed during the Export SHP routine.

When exporting to a DBF file using the Export routine, an individual file will be created for the selected model feature. When exported using the Quick Export routine, a file set is created - one file for each associated data feature. The resulting files may be opened and manipulated using any application which supports the DBF format.

Export DXF (Drawing Exchange Format) Files

GASWorkS provides a routine for exporting data to a DXF file. The User may specify the settings for the export file with the DXF Export Settings screen. The features associated with the screen are described as follows.

DXF Export Settings Screen - *General Tab*

Data Items

Export Coordinates As Flat In Isometric - When selected, the features will be exported as 2D coordinates displayed in a typical isometric configuration.

Include Dimensional Units - When selected, the dimensional units will be included in the exported file. For example, if selected, *Pressure* values will be followed by their assigned pressure unit ("25 Psi").

Include User Graphics - When selected, the User Graphic (Lines and Symbols) features will be included in the exported file.

Include User Text - When selected, the User Text features will be included in the exported file.

Include Variable Identifiers - When selected, the variable identifiers will be included in the exported file. For example, if selected, the *Pressure* values will be preceded by the string "P = ."

Line Type Scale For User Graphic Lines - Specifies the line type scale to use with the User Graphic Lines.

Set Standard Text Style - When selected, the "standard" text style in the DXF will be set to "ARIAL.TTF."

Template Filename - Specifies the Filename of the template file. Enter a Filename or click the *Browse* command button to select a Filename. The use of a template file is optional. This item is only enabled when the *Use Template File* option is selected.

Use Template File - When selected, a template file will be used for the various settings and block and layer definitions. A template file must include certain block and line types in the exported file.

DXF Export Settings Screen - *Pipe Items Tab*

The items on this data tab will only be enabled when the *Include Pipe Items* option is selected.

Data Items

Create Individual Pipe Size Layers - When selected, individual layers will be created in the exported file for each unique pipe Size/Type code.

Export All Pipes As Polyline Pipes - When selected, all of the pipes in the model will be created as polyline pipes in the exported file.

Export Data Text As Multi-Line Text - When selected, pipe text items will be exported as multi-line text in the exported file.

Flow Arrow Size - Specifies the size of the flow arrows created in the exported file.

Include (Data Item)... - When selected, the associated data item will be included in the exported file.

Include Graphics - When selected, the pipe line graphics will be included in the exported file.

Include Pipe Symbols - When selected, the pipe symbols will be included in the exported file.

Include Text For Pipe Elements Only - When selected, only the data text for the pipe elements will be included in the exported file.

Line Pattern Scale - Specifies the scale for the non-continuous line types created in the exported file.

Line Thickness - Specifies the pipe line thickness (width) for the display of the pipe line segments in the exported file. The value is expressed in the length units of the exported file.

Pipe Symbol Size - Specifies the size of the pipe symbols (for non-pipe hydraulic features) in the exported file.

Text Offset - Specifies the offset of the start of the pipe text location in the exported file from the standard GASWorkS determined location. The value is expressed as a percentage of the *Line Thickness* value.

Text Size - Specifies the size of the pipe text items in the exported file.

Use Solid Symbols - When selected, the pipe symbols in the exported file will be displayed as solid symbols.

DXF Export Settings Screen - *Node Items Tab*

The items on this data tab will only be enabled when the *Include Node Items* option is selected.

Data Items

Export Data Text As Multi-Line Text - When selected, the node text items will be exported as multi-line text in the exported file.

Include (Data Item)... - When selected, the associated data item will be included in the exported file.

Include Node Symbols - When selected, the node symbols will be included in the exported file.

Include Text Only For Nodes With Non-Zero Load Values - When selected, the node text items will only be included in the exported file for non-zero load values.

Symbol Size - Specifies the size of the node symbols in the exported file. Node symbols are represented in the DXF file by a block entity. The *Node Symbol Size* value specifies the X- and Y-scale for the block entity.

Text Offset - Specifies the percentage to offset the start of the node text location in the exported file from the node symbol location.

Text Size - Specifies the size of the node text items in the exported file.

GASWorkS™ 10.1 — *Export Routines*

DXF Export Settings Screen - *Customer Items Tab*

The items on this data tab will only be enabled when the *Include Customer Items* option is selected.

Data Items

Include (Data Items)... - When selected, the associated data item will be included in the exported file.

Include Customer Symbols - When selected, the customer symbols will be included in the exported file.

Include Service Lines - When selected, the service lines will be included in the exported file.

Service Line Thickness - Specifies the thickness of the lines used to depict the service lines in the exported file.

Symbol Size - Specifies the size of the customer symbols in the exported file. The value specifies the X- and Y-scale for the block entity.

Text Offset - Specifies the percentage to offset the start location of the customer text items in the exported file from the customer symbol location.

Text Size - Specifies the size of the customer text items in the exported file.

Command Buttons

Browse - Used to find and select a template file name. When selected, displays the File Selection screen. This command button is displayed on the *General* data tab.

Cancel - Closes the screen without saving changes or proceeding with the Export routine.

Clear - Sets all data items to blank (null) values.

Continue - Proceeds with the Export routine.

Open Settings - Opens a previously saved DXF Export Settings file (.dxe). When selected, displays the File Selection screen.

Save Settings - Saves the contents of the currently displayed DXF Export Settings. When clicked, the File Selection screen is displayed.

Notes & Considerations

● The DXF Export routine creates a Drawing Exchange Format (DXF) file. Once created, the file may be imported into or used directly with various CAD or graphics packages.

● GASWorkS attempts to match the colors found in the model with the colors supported in the DXF file. Please be aware that matching is not always exact. Color values may need to be adjusted after the file has been opened in its destination application.

● If the destination application does not read the header section of the DXF file, the layer, block, and application information may not be processed correctly. If this occurs, the imported features may need to be manually moved to the appropriate layers after importing into the destination application.

● If the *Include Extended Entity Data* option is selected, an application named "GW100DATA" will be registered in the DXF file. In addition to the required graphic data, each pipe, node, and customer feature will also include extended entity data, which includes the feature's unique identification. For pipes and customers, the identifier is the Internal ID Number. For nodes, the identifier is the node name.

● Generally, the *Include Graphics* or *Include Symbols* items should always be selected when including Customer, Node, or Pipe Items. For pipes, "Graphics" refers to the graphic lines that show the pipe locations. For nodes, "node symbols" refer to the block symbols that show the node locations. For customers, "customer symbols" refers to the block symbols that show the customer locations.

● If the *Create Individual Pipe Size Layers* option in the *Pipe Items* list is selected, pipes will be grouped on layers representing each unique pipe Size/Type Code. For example, all pipes with a Size/Type Code equal to "2P" will be placed on a layer named "GWPIPE_2P" in the DXF file.

● Select the *Include Valve Nodes* option in the *Pipe Items* list to include valve nodes and their associated number in the DXF file.

Export GASWorkS 9.0 Files

Older versions of GASWorkS model files may be created using either the Export or Quick Export routines. When exporting to a GASWorkS 9.0 file, a set of GASWorkS 9.0 compatible model files is created. Because the older version of GASWorkS does not support all of the data items supported by the newer version, some data loss will occur when exporting to the older version.

Attribute data will not be exported when creating the older files.

GASWorkS™ 10.1   *Export Routines*

Export KML (Keyhole Markup Language) Files

GASWorkS provides a routine for exporting data to a KML file. The settings for the export file may be specified by the User with the KML Export Settings screen. The features associated with the screen are described as follows.

KML Export Settings Screen - *Pipe Items Tab*

The items on this data tab will only be enabled when the *Include Pipe Items* option is selected.

Data Items

Color Assignment - Displays the selected colors to be assigned to the pipes by Size/Type code in the exported file.

Include (Data Items)... - When selected, the associated data item will be included in the exported file.

Pipe Size - Displays selected GASWorkS data items. Only the pipe sizes selected in the *Pipe Size* column will be exported. Any pipe size without a specified color in the adjacent *Color Assignment* column will be exported as "white."

KML Export Settings Screen - *Node Items Tab*

The items on this data tab will only be enabled when the *Include Node Items* option is selected.

Data Items

Include (Data Items)... - When selected, the associated data item will be included in the exported file.

Symbol Size - Specifies the size of the node symbols in the exported file. The *Node Symbol Size* value specifies the X- and Y-scale for the block entity. This item is not currently used.

Text Offset - Specifies the percentage to offset the start of the node text location in the exported file from the node symbol location. This item is not currently used.

Text Size - Specifies the size of the node text items in the exported file. This item is not currently used.

GASWorkS™ 10.1   *Export Routines*

KML Export Settings Screen - *Customer Items Tab*

The items on this data tab will only be enabled when the *Include Customer Items* option is selected.

Data Items

Include (Data Items)... - When selected, the associated data items will be included in the exported file.

Include Service Lines - When selected, the service lines will be included in the exported file.

Service Line Thickness - Specifies the thickness of the lines used to depict the service lines in the exported file.

Symbol Size - Specifies the size of the customer symbols in the exported file. The value specifies the X- and Y-scale for the symbol. This item is not currently used.

Text Offset - Specifies the percentage to offset the start location of the customer text items in the exported file. This item is not currently used.

Text Size - Specifies the size of the customer text items in the exported file. This item is not currently used.

Command Buttons

Cancel - Closes the screen without saving changes or proceeding with the Export routine.

Clear - Sets all data items to blank (null) values.

Continue - Proceeds with the Export routine.

Open Settings - Opens a previously saved KML Export Settings file (.kme). When clicked, the File Selection screen is displayed.

Save Settings - Saves the contents of the currently displayed KML Export Settings. When clicked, the File Selection screen is displayed.

Notes & Considerations

● The KML Export routine creates a Keyhole Markup Language (KML) file. Once created, the file may be imported into mapping and GIS applications that support the format, such as Google Earth.

● GASWorkS provides nine (9) colors for exporting pipe data to a KML file. Nodes and customers are exported using the default symbols. These colors and symbols may be altered later in the target application.

● If an "*Include (data item)...*" option is selected, the associated data item tab will be added to the description of the feature to which it is associated. The exception is the *Include Service Lines* option on the *Customer Items* data tab. This option will create the service lines as a graphic item in the KML file.

● The contents of the exported KML file follow the specifications of the official Open Geospatial Consortium's KML schema.

Export SHP (Shapefile) Files
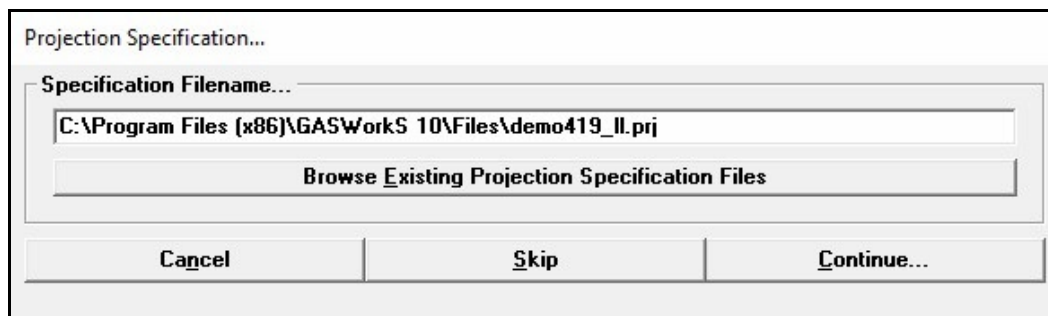
GASWorkS provides a routine for creating shapefile (SHP) files from a GASWorkS model. The routine is available using either the Export or the Quick Export routines.

When exporting to a SHP file using the Export routine, an individual set of shapefiles will be created for the selected model feature. When exported using the Quick Export routine, a group of shapefile sets is created - one set for each associated data feature. The files may be opened and manipulated using any application which supports the ESRI Shapefile format.

The shapefile set includes a geographic file (.shp), an attribute database file (.dbf), and an index file (.shx).

When using the Quick Export routine to create shapefiles, the Projection Specification screen is displayed, requesting the selection of the "projection" file. A projection file defines a set of parameters that define the coordinate system used by the model (usually a state plane coordinate system). If a projection file is assigned, a projection file (.prj) will be included with the shapefile set. The projection file must already exist to be assigned. See the appropriate ESRI documentation for the format and definition of the contents of the projection file.

Projection Specification Screen

Data Items

Specification Filename - Specifies the name of the projection file to use. Either enter a Filename or click the *Browse...* command button.

Command Buttons

Browse - Used to find and select a previously saved projection specification file name. When selected, displays the File Selection screen.

Cancel - Closes the screen without proceeding with the Export routine.

Continue - Proceeds with the Export routine.

Skip - Proceeds with the Export routine without assigning a Projection Specification file.

Export Stoner PD/XY Files

GASWorkS provides a routine for creating Stoner PD/XY files from a GASWorkS model. The PD file will contain various chapters, including a System, NCE, and Node chapter. The XY file will include both a Node and GNode chapter.

Export XLS (Microsoft Excel) Files

GASWorkS provides a routine for creating Microsoft Excel spreadsheet files from a GASWorkS model. The 8.0 format is supported as a single-sheet Workbook format by the Export routine or a multi-sheet Workbook by the Quick Export routine.

When exporting to an Excel 8.0 file using the Export routine, an individual worksheet (".xls" extension) will be created for the associated model feature. The resulting file may be opened and manipulated using any application that supports the associated Excel format.

When exporting to an Excel 8.0 Workbook, a single workbook (".xls" extension) with multiple worksheets will be created. The file may be opened and manipulated using any application which supports the associated Excel format.

MISCELLANEOUS

Calculating Gas Temperatures

GASWorkS supports the modeling of varying gas temperatures in the piping system. When the gas-flowing temperature varies in a system, the temperature change may be calculated along the pipe segments, and the mixed temperature may be calculated at intersections.

Temperature change is calculated based on a User-selected *Heat Exchange* calculation method. Gas cooling due to Pressure Drop (Joule-Thomson Effect) is estimated based on a general rule-of-thumb method and is only valid for gas compositions similar to a "standard" natural gas composition.

Mixing is performed based on a proration of the volumetric flow rate of each stream. The temperature mixing calculation is performed based on upstream flow values. That is, the mixing at a node is based on the temperature of the volumetric flow rates of the pipes flowing into the node. The temperature of the outlet flow leaving the node reflects the mixed value.

Some special requirements must be considered when preparing a model to calculate temperature values and mixing:

● Because the temperature mixing is performed based on upstream flow values, the gas temperature of each supply point, and generally the outlet of regulators and compressors, must be "known." Generally, the *Temperature Known* item is set to "No" for all other node locations. Use the *Temperature Known* item in the Node Data to change the setting.

● The heat exchange data is set for the entire model. This data is accessed using the Solution Data screen. The various values and options are described in the *Data Screen* section. The values must be set appropriately to achieve reasonable results. If actual field values are known, the data values may be adjusted to "calibrate" the model to match the field results.

● Once the data values and known/unknown settings have been correctly set in the model, the *Calculate Temperature Values* solution option and the *Calculate Gas Mixing* solution option must be selected to calculate the *Temperature* values when solving the model.

● When the *Calculate Temperature Values* solution option is selected, the overall heat transfer coefficient is calculated for each pipe segment during the solution. These values are reported with the pipe data.

● The number of iterations required to solve a model using temperature calculation and/or gas mixing will be greater than for a model where these options are not used. The *Maximum Iterations* value on the Solution Data screen may need to be increased to allow convergence.

● The *Show Route Profile* command may be used to graphically display the temperature profile along a selected path.

Change Log

GASWorkS provides a feature for tracking changes to the model. It is known as the Change Log. When Change Log tracking is turned "on," graphic and data changes to the pipe, node, or customer features are recorded in the Change Log. Change Log tracking can be turned "on" or "off" using the *Turn Change Log On/Off* commands. Only changes made using the GDI Edit routines are tracked. Use the *View Change Log* command to view the contents of the Change Log.

The Change Log is useful when documenting corrections and revisions made, especially during the clean-up process of a newly imported model. If the changes are tracked, they can be reported back to the organization responsible for the source data, perhaps to have the source data corrected.

See the *Change Log* topic in the *Graphic Data Interface (GDI)* section for more information on the features.

Command Alias File

The GDI allows User-defined command names to execute a GDI Command on the GDI Command Line. A User-defined command name is known as a command "alias." These values are assigned using the Command Alias File. The Command Alias File is a specially named and formatted ASCII text file that contains the alias and command assignments. The file may be created using any word processing application that supports the ASCII text format. The file must be named "gwcmd.txt" and located in the currently set Initialization File directory (path).

The contents of the file describe the alias to command assignments; one assignment is contained on each line of the file in the following format:

>
> ALIAS, GDI COMMAND LINE NAME
> ALIAS, GDI COMMAND LINE NAME

> *For Example -*

> +PIPE, ADDP
> -PIPE, DELP

The alias can contain most alphanumeric characters except commas (,). The case (upper or lower) of the alias or command name is not significant.

The *Use Command Alias File* preference settings option must be selected to direct GASWorkS to load and use the file. When processing a GDI Command entered on the GDI Command Line, the alias assignments are processed before the standard command names are processed.

Convergence Problems

Maximum Error Bounces From Small To Large

This error can occur for several reasons, the most common of which are described as follows.

● The Row Reduction Limit may be too large - try setting it to zero (0). The effect of the Row Reduction Limit varies depending on the Flow Equation and the *Pressure* values. The Row Reduction Limit is used to limit the pressure adjustment for nodes where the node error is larger than the Row Reduction Limit times the Convergence Tolerance. This helps reduce the over-shoot and under-shoot of the solution. Some equations converge quicker when this reduction is applied to all nodes. When the Row Reduction Limit is set to zero (0), the reduction is applied to all of the nodes.

● The system could be "running out of gas." This refers to the condition when the calculated pressures are less than zero (0). This occurs because the capacity of the system is less than the amount of flow that is trying to be passed through it. If this condition occurs (when it is not supposed to), try decreasing the Design Factor, increasing the pipe efficiencies, or increasing the supply pressures.

Zero Diagonal – Possible Discontinuity From Supply Node(s)

This error generally occurs because a portion of the system is disconnected from the remainder of the system, and the disconnected portion does not contain a node with a "known" pressure. The discontinuity may be the result of a connectivity error, from a turned "off" pipe feature, or from an incorrectly installed regulator or compressor. If this error occurs, try the following to resolve the problem:

>● Note the indicated node Record Number.

>● Use the *Find Node By Record Number* command to locate the affected node.

>● Use the *Trace* command to "Trace All Directions" from the affected node.

>● Follow the highlighted pipe segments to determine the point of discontinuity.

Solution Is Diverging Or Stalled

This error generally occurs because a portion of the system is experiencing negative pressures or a portion of the system is disconnected from the remainder of the system, and the disconnected portion does not contain a node with a "known" pressure.

The negative pressures occur because the demand on the system is greater than the capacity of the system. In this case, use the *System Summary* item from the *Report* menu, with the *System Summary* option selected to determine the total demand on the system (Total Flow Out Of The System). If it is not reasonable, use one of the following to try to find and correct the issue:

- Use the Node Report to review the node load values.

- Use the Data Check routine to search for data errors.

- Use the Design Factor in the Solution Data to reduce the total system demand.

See the previous *Zero Diagonal* topic for suggestions on finding connectivity errors.

Diversified Customer Loads

GASWorkS provides a routine for handling diversified customer loads per the Institution of Gas Engineers Guide (IGE/GL/1). The following section describes how these requirements are implemented in the GASWorkS software.

GASWorkS provides two methods to compute the diversified load of the assigned customers. They are described as follows:

IGE/GL/1 Appendix A5.1.5

This method uses the computational method described in Appendix A5.1.5 to compute the diversified load associated with a combination of varied heating type customers. In the GASWorkS implementation, each customer is assumed to represent a unique group. Using this assumption, the total load for a group of customers may be reduced to:

$$D_{Tot} = \sum a_i + \sqrt{\sum b_i^2}$$

Where a and b are defined by IGE/GL/1.

This value is computed for each node, considering the customers served downstream of the node. All customers along a pipe segment are assumed to be served at the downstream node of the segment.

IGE/GL/1 Appendix A5.1.1-1.4

This method uses the computational methods described in Appendix A5.1.1 through A5.1.4 to compute the diversified load associated with the various heating type customer groups. Using the "fully diversified" equations when the associated customer counts exceed the specified fully diversified limit.

The values are computed for each node, considering the customers served downstream of the node. All customers along a pipe segment are assumed to be served at the downstream node of the segment.

Implementation Notes

Using the above relationship as a foundation, GASWorkS implements diversity into its solution algorithm as follows:

● An initial set of customer counts is calculated for each pipe segment based on the customers assigned to each segment.

● A set of node loads is computed from the pipe customer counts using the relationship shown above.

● The model is solved.

● A new set of customer counts is computed by finding null flow nodes (nodes where all pipe flows are into the node). The flow path is then traced towards the source until a branch or a source is encountered. A new set of pipe customer counts is computed along the flow path by allocating the downstream customer counts to the upstream pipes at each node based on a proration of the incoming pipe flows.

● After the new customer counts are computed for all of the pipe segments, a new set of node flows is computed. The new values are checked against the old values. If each new value is within the Convergence Tolerance of the old value, the diversified flows are "balanced" - the solution and diversity calculation are complete. If any value falls outside the check value, the process is repeated starting at Step 3 until the diversified loads are balanced or the maximum iterations limit is reached.

Model Considerations

The following should be considered when developing a model for the Diversity routine.

● Enter or import the piping network as with any GASWorkS model.

● Customer data features must be present when using the Diversity routine. Use the various "*Add Customer*" commands to enter customers and indicate their supply main. When adding a customer:

● For the *Node Load Application* item, select "Diversified."

● For the *Per Unit Load* item, enter the desired annual heating value. For the *Load Units*, select the desired dimensional units.

● For the *Heating Type* item, select the desired heating type.

● Customers with similar heating types located along a common pipe segment may be represented by a single customer feature by entering the appropriate value for the *Unit Count* item. For example, if there are ten (10) identical customers along a pipe segment, either enter ten (10) individual customer features along the pipe segment or enter one customer with the Unit Count set to ten (10).

● A model may consist of diversified and non-diversified customer loads by appropriately setting the *Node Load Application* method.

● Set the supply pressure values and known/unknown values as with any GASWorkS model. Once the model entry is complete, the model may be solved.

● To use the Diversity routine during the Solution routine, select the desired calculation method from the *Diversity* methods lists in the Solution Data when solving the model. The solution will perform multiple iterations as described previously until a solution is reached or until the maximum iteration limit has been reached. The more complex the piping configuration is, the more diversity iterations are required to complete the solution. Check the last line of the Solution Log to ensure that the solution has successfully been completed.

Extract Routine

GASWorkS allows a portion of an existing model to be extracted into a new model. The Extract routine involves selecting a set of pipes or customers by various means. Once selected, the extracted data will be stored in a new model with a User-specified name. Optionally, the data in the existing model may be deleted once the new model has been created.

The new model will contain the selected pipes and customers (if selected for inclusion). The selected features will be added to the new model. However, any attribute data associated with the original model will not be included in the new model.

Generally, certain data values will need to be set before the extracted model may be solved. Specifically, at least one supply point must be identified (*Pressure Known* is set to "Yes" and *Base Load Known* is set to "No").

Care must be taken when working with an extracted model. Realize that changes made to the extracted model will not be reflected in the original model. If the load (demand) in the extracted model changes, and the supply pressures are taken from the original model, the values may not be appropriate for use with the extracted model.

An extracted model may be added to another existing model using the Append routine in the *File* menu.

If the original data is accidentally deleted during the Extract routine, the *Undo Last Change* command may be used to restore the data.

Either a pipe or customer selection set may be used as the basis for the extraction. If a pipe selection set is used, the connected customers may be optionally included. If a customer selection set is used, only the customers will be extracted. The selection sets may be created using the *Edit Multiple Customers* or *Edit Multiple Pipes* command, or using the Query Specifications routine in the Customer Data Report or Pipe Data Report.

Facility Types

GASWorkS provides a way of grouping or segregating features using a feature called Facility Type. Facility types are similar to layers in many CAD or GIS applications. Facility types are managed using the Facility Settings Editor. The display of an individual Facility may be turned on or off, a Facility may be included or excluded from the Solution and Report routines, and a Facility may be set to allow or prevent editing using the settings in the Editor. Pipe display color, line type, and line width may be controlled by the Facility Type. The Facility Type may be set for pipes, graphics, and text features. Node and customer features inherit their Facility Type from the associated pipe.

Use the *Facility Settings* item from the *Edit* menu to access the Facility Settings Editor. Facility Settings may be saved and opened for other models, and a specific group of settings may be saved as the default for all new models.

Each pipe feature may be assigned a Facility Type. The associated nodes and assigned customers inherit the facility settings of the pipe (main). The *Facility Type* value is intended to allow pipes and customers to be grouped together based on the *Facility Type* value. The Facility Type can represent any group identification. As it is presented in the standard installation of GASWorkS, the Facility Type represents general construction and pressure types. For example, an existing or proposed Facility Type associated with a general operating pressure. The User may revise the contents of the standard Facility Settings as required to meet their unique needs.

Once assigned a Facility Type, the associated pipes may be manipulated using their Facility Type setting. For example, the Mass Update or Report routines can use the Facility Type to create a selection set. The Facility Settings routine may to indicate which pipes may be edited, reported, displayed, or solved. The display style of the pipe lines (color, style, and thickness) may be controlled by the Facility Settings. The Facility Type may also be set and similarly used for User Graphic and User Text features.

The *Active* item identifies which pipes may be edited. It is useful when using automatic manipulation commands such as the *Assign Customer Service Lines* or *Flag Unbroken Intersections*. For example, if a model contains both high-pressure and medium-pressure systems, and all or the majority of the customers are connected to the medium-pressure system, the high-pressure portions of the system could be assigned to the *Main - High Pressure* Facility Type, and the medium-pressure portions could be assigned to the *Main - Medium Pressure* Facility Type. Before executing the *Assign Customer Service Lines* command, the *Main - Medium Pressure* Facility Type could be set to "Active," and the *Main - High Pressure* Facility Type could be set to "Inactive." Only pipes assigned to an "Active" Facility Type may be edited or modified; the *Assign Customer Service Lines* command would only try to assign customers to the "Active" Facility Types; in this case, only pipes assigned to the *Main - Medium Pressure* Facility Type. Similar settings could prevent the *Flag Unbroken Intersections* command from identifying connections between high and medium-pressure portions of the system.

The *Report* and *Display* items may be used to identify which pipe (nodes and customers) features will be included in the various reports and the GDI Display.

The *Solve* item may indicate which pipes to include when the Solution routine is executed. Care must be taken to ensure that a solvable configuration, appropriate known/unknown values, and supply pressures exist when excluding features from the solution.

To control the display style of the pipe lines using the Facility Settings, assign the *Pipe Line Color*, *Pipe Line Style*, or *Pipe Line Thickness* items to the "Facility Table." The values can also be set using the Mass Update routine, the *Edit Multiple Pipes* command, or the individual pipe edit routines.

The Pipe Flow Equation may be controlled using the Facility Type. When the Flow Equation data item is set to "Facility Table" for a pipe feature, the equation associated with the assigned Facility Type is used during hydraulic calculations.

The Pressure Units item in the Facility Settings is not currently used.

There are several special Facility Types associated with certain graphic features. They are:

> *Color Legend* - Used to control the display of the Color Legend text when added as a User Text feature.

> *Bill-Of-Materials Text* - Used to control the display of the Bill-Of-Materials table and text when added as a User Table feature.

> *Marked Views* - Used to control the display of any Marked Views which may be included in the model.

> *Marked Pipe Crossings* - Used to control the display of any Marked Pipe Crossings that may be included in the model.

GASWorkS Limitations

GASWorkS is a very powerful tool for modeling flow in gas piping systems. However, it does have some limitations. The node and pipe limitations are related to the capacity of the particular version purchased. The remaining limitations apply to all versions. The limitations of GASWorkS are described as follows.

Data Types

● A double precision number allows up to 15 significant digits. Unless otherwise specified, values of this type can range from -1.7E+308 to 1.7E+308.

● An integer number allows up to 5 significant digits. Unless otherwise specified, values of this type can range from -32000 to 32000.

● A long integer number allows up to 10 significant digits. Unless otherwise specified, values of this type can range from -2.1E+09 to 2.1E+09.

● A single precision number allows up to 7 significant digits. Unless otherwise specified, values of this type can range from -3.4E+38 to 3.4E+38.

Data Limitations

The following lists the absolute minimum and maximum values allowed by the various parameters used in GASWorkS.

● The node and pipe capacity are version and memory-dependent.

● Node Names must be no more than forty (40) alphanumeric characters in length. Special characters are allowed.

● Pressure values may be any floating point number in the range from minus one atmosphere to the maximum allowed single precision number.

● Node load values may be any valid single precision number.

● Pipe diameter, wall thickness, and roughness values may be any valid positive single precision number.

● Pipe lengths may be any valid positive single precision number.

● The *Maximum Iterations* value must be any valid positive integer value.

● The Design Factor may be any valid single precision number.

● The Convergence Tolerance may be any valid positive single precision number.

● The Upper & Lower Dampening Factors may be any valid positive single precision number.

● The Diameter & Length Path Factors may be any valid positive single precision number.

● Gas Property values may be any valid positive single precision number.

● Temperature values may be any valid single precision number.

● Elevation values may be any valid integer value.

● Node and customer coordinate values must be any valid double precision number from -1E+09 to +1E+09.

● User Text may contain no more than 32,000 characters.

Model Restrictions

● Regulator or compressor-type features cannot be preceded or followed by another regulator or compressor-type feature.

● No more than six (6) pipes may be connected to an individual node.

● At least one node parameter (*Pressure* or *Load* value) must be "known" for each node.

● At least one node pressure value must be known in each supply isolated (independent) portion of the model.

Non-Pipe Hydraulic Features

The majority of most piping systems consist of "pipe" hydraulic features. However, other features can sometimes be found in a piping system - non-pipe hydraulics, including compressors, fittings, regulators, valves, and wells. GASWorkS allows the inclusion of these items in its models in addition to the standard pipe-type features. The following sections describe some considerations that might be useful when including non-pipe hydraulic features in a model.

Compressor

● When a compressor is included in the model, two requirements must be adhered to for the feature to function correctly. The first requirement is that the compressor feature be input so that the From Node represents the upstream (suction) side of the compressor, and the To Node represents the downstream (discharge) side of the compressor. When the compressor symbol is drawn, the large end is the suction side (From Node) of the compressor. If the compressor is input backward, use the *Swap Pipe Ends* command to automatically change the feature's input direction. The second requirement is that a system pressure must be "known" on one side of the compressor.

● During the Solution routine, GASWorkS models a compressor as a "black box." It does not consider any of the characteristics (power rating, etc.) of the compressor during the solution. The software only considers that the feature is a compressor, and as such, knows that the inlet pressure must be less than the outlet pressure.

● If the downstream node pressure is set to "unknown" (*Pressure Known* is set to "No"), the set pressure is used as a check. If the upstream node pressure is greater than the set pressure, the compressor is set to free flow (*Control Style* is set to "Ignore/Bypass"). In this "free flow" mode, the outlet pressure is equivalent to the inlet pressure.

● The compressor operating characteristics are calculated and reported separately from the Solution routine. The characteristics may be viewed using the System Summary Report, the Data Panel by clicking on the associated pipe feature representing the compressor, or by using the *Edit Pipe Data* command.

● See the *Notes & Considerations* section of the *Solution Data* topic in the *Data Screens* section for information on how compressor set pressures are handled during the Solution routine when the *Adjust Compressor Control Pressure* solution option is selected.

Fitting

● For systems where the pipe lengths are generally much longer than the equivalent lengths of the fittings (distribution, gathering, and transmission-type systems), it is not generally beneficial to include the fittings in the model. For systems like these, the inclusion of fittings unnecessarily complicates the model while providing only a very little increase in the model's accuracy.

● For systems where the pipe lengths are generally comparable to the equivalent lengths of the fittings (fuel piping, meter sets, regulator sets, and yard piping systems), it may be beneficial to include the fittings in the model. For systems like these, the inclusion of fittings does tend to complicate the model; however, in this case, their inclusion may provide a significant increase in the model's accuracy.

● Inclusion of fittings can sometimes cause the number of iterations required to solve the model to increase. This occurs because, generally, the pressure drop across fittings is quite small, and features with small pressure drops tend to take longer to solve or balance.

● An alternative to including *Fitting* Hydraulic Types in a model is to include the equivalent length of the fittings in the hydraulic length of the associated pipe segments. Or, attach fittings directly to the associated pipe segment. The latter method is the easiest way to handle fittings.

Regulator

● When a regulator is included in a model, the regulator-type feature must be input so that the From Node is the upstream (high-pressure) side of the regulator, and the To Node is the downstream (low-pressure) side of the regulator. When the regulator symbol is drawn, the upstream (From Node) side of the regulator is always shown to the left of the "control head" when the head is pointing up. If the feature is input backward, use the *Swap Pipe Ends* command to automatically change the feature's input direction.

● GASWorkS models regulators in two different manners:

     ● If the downstream node pressure is set to "known" (*Pressure Known* is set to "Yes"), the regulator is modeled as a "black box." It does not consider any of the characteristics (valve coefficient, etc.) of the regulator during the solution. In this case, the software only considers that the feature is a regulator and, as such, knows that the inlet pressure must be greater than the outlet pressure - the outlet pressure is also maintained at the set pressure.

     ● If the downstream node pressure is set to "unknown" (*Pressure Known* is set to "No"), the regulator is still modeled as a "black box." However, the downstream pressure can change depending on the upstream and downstream conditions. In this case, the downstream or outlet pressure is not necessarily maintained at the set pressure.

● See the *Notes & Considerations* section of the *Solution Data* topic in the *Data Screens* section for information on how compressor set pressures are handled during the Solution routine when the *Adjust Regulator Outlet Pressure* solution option is selected.

Valve

● For systems where the pipe lengths are generally much longer than the equivalent lengths of the valves (distribution, gathering, and transmission-type systems), it is not generally beneficial to include the valves in the model. For systems like these, the inclusion of valves unnecessarily complicates the model while providing only a very little increase in the model's accuracy.

● For systems where the pipe lengths are generally comparable to the equivalent lengths of the valves (fuel piping, meter sets, regulator sets, and yard piping systems), it may be beneficial to include the valves in the model. For systems like these, the inclusion of the valves does tend to complicate the model; however, in this case, their inclusion may provide a significant increase in the model's accuracy.

● Inclusion of valves can sometimes cause the number of iterations required to solve the model to increase. This occurs because, generally, the pressure drop across valves (especially valves the same size as the associated pipe) is quite small, and features with small pressure drops tend to take longer to solve or balance.

● In the Pipe Data, use the *Percent Open* item to account for partially open valves. Set the *Status* to "off" to turn a valve completely off. Note that any flow feature may be turned "off" using the *Status* item.

● An alternative to including valves as a hydraulic feature in a model is to include the equivalent length of the valves in the hydraulic length of the associated pipe segments. Or, attach the valve as a fitting directly to the associated pipe segment. The latter method is the easiest way to handle valves.

● For tracing purposes, an alternative to including valves as hydraulic elements in the model is to include them as valve nodes. Valve nodes are not included in the flow (hydraulic) model, but they may be used to design and check emergency isolation areas.

Well

● GASWorkS allows the inclusion of wells as a Hydraulic Type, provided that the well performance characteristics are available. To include a well-type feature as a Hydraulic Type, the equation coefficient and exponent must be calculated from a multi-point flow test of the well. This data is not always available, and if it is available, it is generally only valid for the earlier life of the well. Unless retested, this data is seldom valid for a mature well.

● The advantage of including a well-type feature in a model is that the well's flow rate will behave more realistically in the model. As the downstream system pressure increases, the well flow decreases. And as the downstream system pressure decreases, the well flow increases.

● As an alternative to including wells as Hydraulic Types in the model, a well may be modeled as a node with a known supply (positive load value).

Personalize GASWorkS

GASWorkS attempts to accommodate personal preferences by allowing the User to make various preference and behavior settings. Some settings are applied only to the GASWorkS operating environment; others are applied and saved with the model data. The following table lists some of these settings and whether they apply to the GASWorkS operating environment or to the model data. Not all of the settings and options are included in the table. Descriptions of other options are included with the screen descriptions that they are associated with.

Settings/Options	*Description*	*Saved With*
Path (File) Locations	Sets the location where the GASWorkS initially looks for various files. These settings may be used to set a default location for model files, initialization files, Property Tables, and support files. ①	GASWorkS Operating Environment
Property Tables	Sets the location where GASWorkS looks for the assigned Property Tables.	GASWorkS Operating Environment
Preference Settings	Allows various User Interface behaviors to be selected and set by the User.	GASWorkS Operating Environment
Report Options	Allows various settings affecting the report behaviors to be selected and set by the User.	Model Data
Graphic Settings ②	Allows various settings affecting the graphical display of the various model features to be selected and set by the User.	Model Data
Solutions Settings	Allows various settings affecting the solution behavior to be selected and set by the User.	Model Data
Default Data Values	Allows various data values, dimensional units, and settings to be selected and set by the User.	Model Data

Notes

① Data saved with the GASWorkS Operating Environment are saved to the gw10.ini file in the GASWorkS "application" directory (folder).

② These settings may be saved as the "default data," in which case they are used similarly to the data saved in the GASWorkS Operating Environment.

Pipe Sizing

GASWorkS provides a feature for determining the required pipe sizes based on certain User-defined criteria. The process involved in finding these sizes is referred to as optimization. The optimization process involves three separate steps.

The first two steps involve identifying which pipes are to be sized in the model and identifying which pipe sizes and types may be used from the Pipe Property Table. The third step, the actual Pipe Sizing routine, is implemented as part of the Solution routine. When the *Solve & Calculate Pipe Sizes* command button is clicked, the Solution routine will attempt to minimize the size or cost of the system being modeled by attempting to find the least size or least cost combination of pipes that will satisfy the User-specified pressure requirements. Pipe sizes and cost values are extracted from the Pipe Property Table. Only pipe sizes and types in the Pipe Property Table, where the *Use When Sizing* item is set to "Yes," will be considered or used during the optimization. In the model, only pipes where the *Allow Sizing* item is set to "Yes" will be sized.

The System Summary Report compliments the Pipe Sizing routine by allowing the cost and size totals to be quickly estimated and compared between varying design scenarios. By comparing the total cost of the various designs, the most cost-effective design can easily be identified. The cost values are calculated by multiplying the pipe *Hydraulic Length* value by the *Cost Per Length* value contained in the Pipe Property Table. For example, the cost of a pipe segment 1000 feet long with a unit cost of 8 would be 8 times 1000 or 8000.

The Pipe Sizing routine may be used to size an entire system or only a single pipe segment. It is especially useful when designing a new system or designing replacement or renewal facilities for an existing system. The Pipe Sizing routine is iterative. The number of iterations required to complete the process depends on the number of pipe segments identified for inclusion in the Pipe Sizing routine and the number of allowable pipe sizes that may be used.

As part of the optimization setup, the User establishes certain conditions they wish to maintain. The Pipe Sizing routine attempts to maintain those conditions while minimizing the pipe sizes or costs of the system. Various types of conditions may be set:

- A pressure limit may be established for the entire system.

- A pressure limit may be established at certain node locations (referred to as "Condition Nodes"). Up to ten (10) locations may be specified, each with a different pressure value.

- The pressure limits may be set to either maximum or minimum values.

- A *Maximum Velocity* value may be established.

For example, consider a system that contains an industrial customer who has a minimum inlet pressure requirement of 15 psig. The node representing that customer could be identified as a *Condition Node*, and the *Condition Pressure* would be set to 15. The system pressure value is used to establish an overall limiting system pressure. This value specifies that no pressure in the system may be higher or lower than the specified system pressure, depending on whether the pressure represents maximum or minimum values. If a *Condition Node* is not identified, only the overall system pressure is used when sizing the system.

As an example of how these two conditions are used in the Pipe Sizing routine, consider the industrial customer described above. In addition to the customer's pressure requirements, let's add the condition that the minimum system pressure is 10 psig. The Pipe Sizing routine would attempt to find the minimum size pipes that could supply at least 15 psig to the industrial customer while ensuring that no other pressure in the system is less than 10 psig.

In all cases, the velocity would be limited to, at, or below the specified limit value.

The general process for determining pipe sizes is as follows:

● If the *Reset Diameters To Minimum Size* solution option is selected, all of the pipes identified for sizing will be set to the minimum allowed pipe size. Otherwise, the current pipe sizes will be used as the basis for starting the optimization.

● The model is solved, and the condition pressures are checked until the conditions are satisfied. During the check process, if the current value is less than the specified value for minimum limit values or greater than the specified value for maximum limit values, the specified *Path Processing* method is used to identify which pipes to modify.

● The size of the identified pipes is increased one size. The model solution and check process is continued until all of the condition pressures are satisfied. Once the *Condition Pressure* conditions are satisfied, a similar "solve and check" process is used to check the system pressures.

● The velocity values are checked, and the pipe sizes are adjusted accordingly.

● The previous steps result in an acceptable set of pipe sizes, meeting the various sizing requirements. A final review of the preliminary results is performed to determine if any of the resized pipes may be downsized while still maintaining the specified sizing requirements. Changes are made accordingly until the final set of pipe sizes is selected.

The key to the Pipe Sizing routine is to determine which pipe segments to change. The pipe selection process is controlled by the selected *Path Processing* method.

The various processing methods are described in the following table:

Path Processing Method	*Description*
Alternate Processing	This method alternates using the smallest and largest flow processing methods.
Flow-Pressure Drop Processing	This method uses pipe flow rates to determine the "path" and then uses a ratio of the pressure drop and pipe length to determine which segments to modify. This method generally results in a graduated size "trunk" and "branch" type system that most designers are accustomed to. This is the recommended processing method. The *Flow-Pressure Drop Processing* method may be "tuned" using the *Pressure Drop Path Factor* and *Length Path Factor* values on the *Other Settings* data tab on the Solution Data screen. Raising or lowering one of the values causes it to have more or less influence on the "resistance" portion of the processing routine. When computing the pressure drop/pipe length ratio, the pressure drop and pipe lengths are raised to their respective "factors." The product of the flow rate and the ratio is used to select the pipes to be modified.
Flow Processing Only	This method uses the pipe flow rates to determine the "path" and then uses the pipe "resistance" values to determine which segments to modify. This method generally results in a graduated size "trunk" and "branch" type system that most designers are accustomed to. The *Flow Processing Only* method may be "tuned" using the *Diameter Path Factor* and *Length Path Factor* values on the *Other Settings* data tab on the Solution Data screen. Raising or lowering one of the values causes it to have more or less influence on the "resistance" portion of the processing routine. The pipe resistance, as used by this routine, is represented by the ratio of the length to the diameter, both raised to their respective "factors." The product of the flow rate and resistance is used to select the pipes to be modified.
No Path Processing	This method ignores flow relationships and focuses on finding the pipes with the highest pressure drop. These pipes are then identified for change. The process assumes that any of the pipes in the model can affect the pressure at any of the nodes.

Path Processing Method	Description
Process Along Largest Flow	This method finds the upstream path from the check location with the largest flow rates and finds the pipes with the highest pressure drop. These pipes are then identified for change.
Process Along Smallest Flow	This method finds the upstream path from the check location with the lowest flow rates and finds the pipes with the highest pressure drop. These pipes are then identified for change.

The sizing routine allows limits to be set for the velocity of the pipes in the system. The maximum velocity that may be physically achieved in a pipe segment is equivalent to the speed of sound under the conditions found in the pipe segment. When sonic velocity is achieved, the flow is said to be "choked." At choked flow, the flow rate cannot be increased without an increase in inlet pressure (an additional lowering of the outlet pressure does not increase the flow rate). This is similar to what happens in a regulator at "critical" flow. The most common case of choked flow in a gas piping system is found in a flare or relief valve stack piping. Chocked flow is rarely achieved in other portions of a normal gas piping system.

It would, however, make sense that there needs to be some practical limit for gas velocity in design calculations. However, there seem to be many opinions on an appropriate velocity for gas flow. Values range from 20 Ft/Sec to about 300 Ft/Sec or more. There are two general criteria cited for limiting velocity. One is to reduce noise levels in above-ground piping. Two, it is to reduce the movement and abrasion caused by dust and debris in the piping system. It is up to the User to select an appropriate limit; however, inappropriately limiting the velocity increases pipe size and cost of a system without providing any real hydraulic design benefits.

During the Pipe Sizing routine, only *Pipe* Hydraulic Type features where the *Use When Sizing* option is set to "Yes" will be considered for a new pipe size. The size of the fitting and valve-type features have not changed. It is best to exclude these features from the model using the Pipe Sizing routine. If they are included, they will need to be manually changed as appropriate after the Pipe Sizing routine is complete. The Pipe Sizing routine may need to be re-executed after changing them.

The Pipe Sizing routine provides a very powerful tool for the system designer. However, it cannot replace the intuition, knowledge, and discretion of an experienced designer. The results of the Pipe Sizing routine will provide a good start to the design process. However, these results should not be considered to be a "final design." The pipe sizing results should always be reviewed and revised as required to complete the design process.

There are additional values and settings that can also affect the sizing routine:

> *Facility Settings* - This item can be used to identify which pipes are to be considered for sizing. Depending on the value of the Facility Type selection in the Solution Data - Pipe Sizing Control data, only pipes assigned to the specified Facility Type will be considered for sizing.

Sizing Group - This item indicates the data list used to identify the sizing groups. Several example lists are provided when the software is installed. The list is selected using the *Property Tables* item from the *Preferences* submenu of the *File* menu.

Pipe Sizing Table - This item can be used to identify which pipe Property Table to use for the source of available pipe sizes. It can be the same or different than the currently assigned Pipe Property table. The Pipe Sizing Table is in the Solution Data - Pipe Sizing Control data.

Query Features

The Mass Update routine is controlled, the content of the Data Reports is modified, and the GDI Image may be color-coded or items may be "flagged" using a User-defined query specifications. A query specification is merely a means of telling GASWorkS which records to display or modify. Only records that meet the specifications are used, while records that do not match the specifications are ignored. When used with the Mass Update or Color Coding routine, the query specifications tells GASWorkS which records to change. When used with the Data Reports, the query specifications tells GASWorkS which records to display in the report.

The specifications may be saved and opened for later use. When used with the Data Reports, multiple queries may be sequentially implemented.

The following table summarizes the GASWorkS query features.

Operator	Definition	Example	Applies To
= (Equal To)	Finds all records with a *Where* item equal to the *To* value.	**Pipe Size/Type Code = 4P** Finds all pipe records where the *Size/Type Code* equals "4P."	Numeric & String values
< > (Not Equal To)	Finds all records with a *Where* item not equal to the *To* value.	**Pipe Size/Type < > 4P** Finds all pipe records where the *Size/Type Code* does not equal to "4P."	Numeric & String values
< (Less Than)	Finds all records with a *Where* item less than the *To* value.	**Node Pressure < 10** Finds all node records where the *Node Pressure* value is less than 10.	Numeric values
> (Greater Than)	Finds all records with a *Where* item greater than the *To* value.	**Node Pressure > 10** Finds all node records where the *Node Pressure* value is greater than 10.	Numeric values
<= (Less Than Or Equal To)	Finds all records with a *Where* item less than or equal to the *To* value.	**Node Pressure <= 10** Finds all node records where the *Node Pressure* value is less than or equal to 10.	Numeric values
>= (Greater Than Or Equal To)	Finds all records with a *Where* item greater than or equal to the *To* value.	**Node Pressure >= 10** Finds all node records where the *Node Pressure* value is greater than or equal to 10.	Numeric values

Operator	*Definition*	*Example*	*Applies To*
Like	Finds all records with a *Where* item that contains the specified *To* value.	**Node Name LIKE Reg** Finds all node records where the *Name* item contains the string "Reg."	String values
Not Like	Finds all records that do not contain the specified *To* value.	**Customer Link ID Number NOT LIKE Commercial** Finds all customer records where the *Link ID Number* item does not contain the string "Commercial."	String values
Min	Finds a specified number of records that possess the lowest *To* values.	**Node Pressure MIN 10** Finds the node records with the ten lowest pressure values.	Numeric values
Max	Finds a specified number of records that possess the highest *To* values.	**Node Pressure MAX 10** Finds the node records with the ten highest pressure values.	Numeric values
Starts With	Finds all records that start with the specified *To* value.	**Node Name STARTS WITH HP** Finds all node records where the node *Name* starts with the string "HP."	String values
Ends With	Finds all records that end with the specified *To* value.	**Pipe Size/Type Code ENDS WITH PE** Finds all pipe records where the Size/Type Code ends with the string "PE."	String Values
Note - Not all operators may be used with all data items.			

Working With Gas Properties

When different supply sources have different gas property values, GASWorkS allows the gas properties from the different flow streams to be mixed at intersections. Only the gas properties (Specific Gravity, Viscosity, Heating Value, and Specific Heat Ratio) are mixed. Mixing of the gas composition is not supported.

Mixing is performed based on a proration of the volumetric flow rate of each stream. The gas mixing calculation is performed based on upstream flow values. That is, the gas mixing at a node is based on the properties of the volumetric flow rates of the pipes flowing into a node. The gas properties of the outlet flow leaving the node reflect the mixed value.

Some special requirements must be considered when preparing a model to use gas property mixing:

● Because the gas mixing is performed based on upstream flow rate values, the gas properties of each supply point must be "known." Generally, the *Properties Known* item is set to "No" for all other node locations. Use the Gas *Properties Known* item in the Node Data to change the setting.

● When gas mixing is used, each node with "known" gas properties must be at the terminal end of a one-way segment.

● The gas properties for known locations may be calculated from a known gas composition using the *Calculate Gas Properties* item from the *Analysis* menu.

● Once the data values and known/unknown settings have been correctly set in the model, the *Calculate Gas Mixing* solution option must be selected to calculate the mixed gas property values when solving the model.

● If a Compressibility Method is used, a gas properties file for the model must be created. When the gas composition varies throughout the system, as when mixing is used, the gas properties file should represent an "average" composition for the gas in the system. The gas properties file must be named the same as the model. The file may be created using the Gas Properties Calculator screen.

● The number of iterations required to solve a model using gas mixing and/or calculating compressibility will be greater than for a model where these options are not used. The *Maximum Iterations* value may need to be increased to allow convergence.

Working With The GeoMap Functions

GASWorkS allows geographic map services, such as those from Google and Microsoft, through its GeoMap functions. The GeoMap functions allow roadway and satellite images from these services to be displayed as a background to a GASWorkS model. The geolocating function allows a location to be found by address or geographic name. These functions may be helpful when building or maintaining a model. The following provides some details and considerations for working with and using the GeoMap functions.

The GeoMap function is only available in certain versions of GASWorkS.

An Internet Connection

There must be a continuous internet connection to have full use of the GeoMap functions. The displayed images are downloaded directly from the map service via an internet connection. The geolocation function requires internet access to the map service's geolocating "engine." The execution speed of the GeoMap functions will depend on the speed and quality of the internet connection being used.

The Map Service

The "map service" provides the images for display and the engine for the geolocating service. The following services are supported:

> Google Maps™
> Microsoft Bing Maps™
> OpenStreetMap

In most cases, each map service uses different imagery of differing ages and resolutions. For a specific area, one service may be more up-to-date than another. It may be necessary to use different services in the same model to get the desired coverage. Coverage may not be available for all areas.

These map services are provided under license from the service provider. You must agree to their terms of use and privacy policy to use the GeoMap functions. Per the providers' requirements, service and imagery copyright information is displayed along with the image. ***Access to the map service is controlled by the service provider (Licensor) no guarantee is made or implied that access to the service will be available or that the content will be of a useable quality.***

Depending on the service selected, different features will be available for display, including satellite imagery, roadways, or a combination of roadways and imagery.

Only one service may be used at a time.

A detailed description of the terms-of-use and privacy policy for each of the map service providers may be viewed by selecting the appropriate item from the *Third Party Terms & Conditions* submenu of the *Help* menu.

The Coordinate System

To properly display a model over a GeoMap background image, the model must use the Web Mercator projection (WMP). The WMP uses the World Coordinate System (WCS) values of Longitude and Latitude to identify locations. In the WMP, locations with a latitude further than about 85° north or south from the Equator are not supported.

In GASWorkS, the WMP is supported by the "Web L/L" dimensional unit. Location values must be entered in decimal degrees of Longitude (X) and Latitude (Y). Note that the coordinate entry order is reversed from the customary Latitude and Longitude convention. Generally, values to six decimals are required to accurately define a location.

If a model is using a plane coordinate system, it will need to be re-projected into the WCS before it may be used with the GeoMap functions. If the plane coordinate system is known, and an appropriate projection file is available, the model coordinates may be "re-projected" into WCS values using the *Project Coordinates* command. Once re-projected, it may be displayed using the Web L/L coordinate units.

The WCS measures latitude as degrees north or south from the Equator, with southern locations being a negative value. Longitudes are measured as degrees east or west from the Greenwich Meridian (Greenwich, England), with western locations being a negative value. Latitudes range from -90° to 90°, with 0° representing the Equator. Longitudes range from -180° to 180°, with 0° representing the Greenwich Meridian.

Finding A Location

The GeoMap functions include a geolocation function, which may be used to find a location by address or geographic name. To use the function, enter **GEOFIND** on the GDI Command Line. If the entered location is found, a flag will be placed, and the GDI Image will be zoomed to that location. Enter **GEOCLEAR** on the GDI Command Line to remove the flag.

Windows 7 or newer is required to use the Google geolocating engine.

Zooming

When the GeoMap functions are enabled, the *Zoom To Fit* command resizes the GDI Image to display all model features. If no features are entered yet, the GDI Image will be resized to display the extent of the entire world. To display the entire world at other times, enter **WORLD** on the GDI Command Line. The *Zoom* and *Pan* commands work as normal when the GeoMap functions are enabled.

The maximum supported zoom level will vary depending on the map service being used. If you try to zoom in further than what is supported by the service, the GeoMap background image will not be displayed; however, the model data will continue to be displayed.

Building A New Model - By Tracing

A GeoMap background may be used as a base map or reference for building a new model. Let's consider that you would like to build a model by manually tracing over a GeoMap background image. Following are the general steps required to use a GeoMap background image for this purpose:

• Create a new model using the *New* item or *New Project* item from the *File* menu.

• If not enabled, enable the GeoMap functions by typing **GEOMAP** on the GDI Command Line.

• Select the **Web L/L** dimensional units for the *Coordinates* item on the Default Data Values screen using the *Set Defaults* item from the *Utilities* menu.

• Attach the GeoMap background as a new image on the Background Image Settings screen using the *Background Settings* item from the *Graphics* menu.

• Zoom to the area where you would like to work. One way to do that is to use the *Find Geographic Location* command to flag the location, then use the *Zoom* and *Pan* commands to reach that location. To execute the *Find Geographic Location* command, type **GEOFIND** on the GDI Command Line.

• Once you have reached the area where you would like to work, use the model construction and edit commands to build the model.

See the GASWorkS Help Guide for more detailed instructions.

Building A New Model - By Importing Data

A GeoMap background can also be used as a base map or reference for a model built by importing data. Let's consider that you would like to build a model by importing piping data from a shapefile and then overlay the model data on a GeoMap background. Following are the general steps required to use a GeoMap background image for this purpose:

    • Use the *Import* item from the *Utilities* menu to import a "pipe" shapefile. Set the appropriate coordinate units for both the source and model before finishing the Import routine.

    • If the model Coordinates units are not in WCS values, merge additional data such as junction, customer, or service locations, then use the *Project Coordinates* command from the GDI Command List to re-project the model to WCS values.

    • If not already enabled, enable the GeoMap functions by typing **GEOMAP** on the GDI Command Line.

    • Select the **Web L/L** dimensional units for the *Coordinates* item on the Default Data Values screen using the *Set Defaults* item from the *Utilities* menu.

    • Attach the GeoMap background as a new image on the Background Image Settings screen using the *Background Settings* item from the *Graphics* menu.

    • Display the entire model using the *Zoom To Fit* command.

    • Use the model checking, editing, and construction commands to complete the model.

See the Help Guide for more detailed instructions.

Other Background Images

Other background images may be used with the GeoMap background - vector style images in a Geographic Information System (GIS) format work best. For example, shapefile (.shp) or Keyhole Markup Language (.kml) files work well. To use these images, their coordinates must be in WCS values. Generally, KML files are already in WCS values. A shapefile may or may not be in WCS values. If the file is not in WCS values, an appropriate projection file (.prj) is required to use the image with the GeoMap background. An option for projecting a shapefile background image into the required coordinate system is provided in the Background Image Settings; select it as appropriate.

Other vector-based images may be used; however, they must be projected into the WCS values before being attached.

Raster images may be used with the GeoMap background; however, the scale and insertion position must be appropriately set. This may take a few iterations to get the desired display. Raster images will "block out" the underlying GeoMap background.

Sizes & Settings

Some GASWorkS parameter sizes and settings may need to be adjusted when working in Web L/L (and Degree L/L) coordinate units. Any value that lists its dimensional unit as "coordinate units" will need its value entered in decimal degrees. Most of these items can also be set to an alternative unit, such as "% Of Display Width" for symbol sizes or "Printers Points" for font sizes. It might be easiest to use the alternative unit in these cases. If a decimal degree value is used, for reference, 0.00003° of latitude or longitude is approximately equal to 10 feet or about 3 metres in length near the equator at sea level.

Default feature colors, symbols, and line sizes may need to be adjusted to make them more visible when a GeoMap Background image is displayed.

Valve Nodes

GASWorkS allows valves to be considered using various methods. Valves may be included in the model as a *Valve* or a *Fitting* Hydraulic Type feature, as an attached fitting, or as a valve node. A valve node is associated with a node feature but is not considered a part of the hydraulic model. Valve nodes are used only for display and when performing a "valve" style Trace.

In most cases, the additional restriction associated with a valve may be effectively accounted for by attaching a "valve" type fitting directly to the pipe segment. Generally, where the pipe lengths are long compared to the equivalent length of the valves, the omission of system valves in the actual model data has a negligible influence on the results of the analysis. Because of this, system valves are often left out of the hydraulic model. However, it might be desirable to keep track of a valve's location for non-modeling purposes (such as tracing) - this is the purpose and designed use of the valve node features.

The model creation process may be greatly simplified by maintaining valves as valve nodes, as opposed to including them as actual flow elements in the model.

The location of a valve node is displayed using an open circle with an "X" in it. The Hydraulic Type (piping) symbols are displayed in a common size.

APPENDIX

Atmospheric Pressure Methods

As used by GASWorkS, the atmospheric pressure refers to the average pressure that the atmosphere exerts on an object at a given elevation. It is similar to barometric pressure, but not the same. As used in most analyses in the gas industry, the "atmospheric pressure" represents the pressure associated with a "standard" atmosphere, not the actual atmosphere (which changes constantly). The atmosphere extends from the surface of the Earth to the edge of outer space. The approximate depth (or height) of the Earth's full atmosphere is about 60 miles. However, in terms of atmospheric pressure, it can be considered to only extend about 10.5 miles (about 55,500 feet). Above this height, the atmospheric pressure is considered to be zero. At sea level, the atmospheric pressure is about 14.7 psia; this is the pressure exerted on an object at sea level. It represents the weight of the 10.5 miles of atmosphere above the object. As an object is elevated, the amount of atmosphere above it, and therefore the weight upon it is reduced. Because the weight of the atmosphere upon the object at its new elevation is reduced, the amount of pressure exerted on the elevated object is also reduced.

The same holds for changes in the geographic elevation of a gas piping system. As the elevation of the system increases, the atmospheric pressure decreases. Gas is a compressible fluid; as such, its physical properties are affected by the amount of pressure exerted on it. As an example, a cubic foot of gas collected in a balloon at 6,000 feet will contract to a volume less than the original one cubic foot when transported to a lower elevation. Why? Because it is a compressible fluid and at the decreased elevation, the pressure exerted on the balloon is greater than its original collection elevation, compressing the gas in the balloon. The inverse holds true for an increase in elevation. If the balloon were elevated, the original cubic foot of gas would expand because of decreased atmospheric pressure at the new, higher elevation.

Because changes in atmospheric pressure affect the properties of the gas and the absolute pressure values, they need to be compensated for when modeling a system. GASWorkS uses the *Node Elevation* values to calculate the atmospheric pressure for each node. Where significant elevation changes are present in the system, the elevation for each node should be determined and applied. Where elevation changes are insignificant, an average elevation may be determined and applied to all nodes.

Unfortunately, several methods are available to estimate the atmospheric pressure at elevated heights above sea level. Each method provides different values and disagrees on the value at sea level. Fortunately, in most pipe flow calculations, the actual absolute value of the atmospheric pressure generally is not critical, so long as the relative differences between the various elevation values are representative. The atmospheric pressure equations supported by GASWorkS are described as follows.

AGA

As implemented by GASCalc, the atmospheric pressure calculated using this method uses the following equation. The equation was derived from the cited reference.

$$P_{ATM} = \left[\frac{55457 - ELEV}{54735 + ELEV} \right] \times 14.54$$

Reference 1

ASHRAE 2009

As implemented by GASCalc, the atmospheric pressure calculated using this method uses the following equation. The equation was derived from the cited reference.

$$P_{ATM} = 14.696 \times (1 - 0.0000068754 \times ELEV)^{5.2559}$$

Reference 4

Handbook Of Chemistry & Physics

As implemented by GASCalc, the atmospheric pressure calculated using this method uses the following equation. The equation was derived from the cited reference.

$$P_{ATM} = 14.69 \times \left[\frac{(288.15 - 0.0019811 \times ELEV)}{288.15} \right]^{5.255877}$$

Reference 2

ISHM

As implemented by GASCalc, the atmospheric pressure calculated using this method uses the following equation. The equation was derived from the cited reference.

$$P_{ATM} = \left[\frac{56267 - ELEV}{55545 + ELEV} \right] \times 14.55$$

Reference 3

NOAA 1976

As implemented by GASCalc, the atmospheric pressure calculated using this method uses the following equation. The equation was derived from the cited reference.

$$P_{ATM} = 14.696 \times (1 - 0.00000686 \times ELEV)^{5.2554}$$

Reference 5

Equation Variables

Unless otherwise noted, the following base units and variable definitions apply to all the methods and equations described in the *Atmospheric Pressure Method* section.

E_1 = Pipe Inlet (Upstream) Elevation, feet
E_2 = Pipe Outlet (Downstream) Elevation, feet
ELEV = Average Elevation $(E_1 + E_2)/2$, feet
P_{ATM} = Average Atmospheric Pressure, psia

References

1) American Gas Association, *Measurement*, GEOP Series Book M-1, 1993.
2) CRC Press, *CRC Handbook Of Chemistry & Physics*, 73rd Edition, 1992.
3) International School Of Hydrocarbon Measurement, *Practical Methods*, Report Number 4, 1976.
4) American Society of Heating, Refrigerating and Air-Conditioning Engineers, *Fundamentals,* 2009 ASHRAE Handbook Inch Pound Edition.
5) National Oceanic and Atmospheric Administration, *US Standard Atmosphere*, 1976. As cited in American Gas Association, Measurement of Natural Gas by Turbine Meters, Report No 7, 2006.

Compressibility Methods

Natural gas mixtures represent a "real" gas and deviate from the ideal gas laws. This deviation becomes more pronounced at elevated pressures. The Compressibility Factor is used to accommodate for these deviations. GASWorkS allows the selection of one of several methods to calculate the Compressibility Factor when applicable. These support equations are described in the following sections.

AGA8-92-Detailed

This method uses the procedures outlined in American Gas Association Report No. 8, "Compressibility Factors of Natural Gas and Other Related Hydrocarbon Gases," Second Edition, November 1992, Second Printing July 1994 for the "Detailed Characterization Method." This method is based on the mole fraction values of each component in the associated gas composition. The full details of this method are too extensive to include here; refer to Reference 1 for further information on the calculation of the Compressibility Factor using this method.

To use this method, a common gas properties file must be assigned to the model. Select the *Calculate Gas Properties* item from the *Analysis* menu to create and save the required gas property file. This method requires a complete description of all of the components found in the gas mixture.

AGA8-92-Gross-1

This method uses the procedures outlined in American Gas Association Report No. 8, "Compressibility Factors of Natural Gas and Other Related Hydrocarbon Gases," Second Edition, November 1992, Second Printing July 1994 for the "Gross Characterization Method - Method 1." This method is based on the volumetric gross heating value, relative density (specific gravity) value, and the mole fraction of the carbon dioxide component of the associated gas composition. The full details of this method are too extensive to include here; refer to Reference 1 for complete information on the calculation of the Compressibility Factor using this method.

To use this method, a common gas properties file must be assigned to the model. Select the *Calculate Gas Properties* item from the *Analysis* menu to create and save the required gas properties file. This method requires that the Heating Value, Specific Gravity, and the Carbon Dioxide (CO_2) component of the gas mixture be identified.

GASWorkS™ 10.1

AGA8-92-Gross-2

This method uses the procedures outlined in American Gas Association Report No. 8, "Compressibility Factors of Natural Gas and Other Related Hydrocarbon Gases," Second Edition, November 1992, Second Printing July 1994 for the "Gross Characterization Method - Method 2." This method is based on the relative density (specific gravity) value and the mole fractions of the carbon dioxide and nitrogen components of the associated gas composition. The full details of this method are too extensive to include here; refer to Reference 1 for complete information on the calculation of the Compressibility Factor using this method.

To use this method, a common gas properties file must be assigned to the model. Select the *Calculate Gas Properties* item from the *Analysis* menu to create and save the required gas properties file. This method requires that Specific Gravity and the Carbon Dioxide (CO2) and Nitrogen (N2) components of the gas mixture be identified.

GPSA

The Gas Processors Suppliers Association (GPSA) compressibility method used by GASWorkS is calculated as follows...

$$P_{AVE} = \frac{2}{3} \times \left(P_1 + P_2 - \frac{P_1 \times P_2}{P_1 + P_2} \right)$$

$$\text{For } P_{AVE} > 100, \qquad Z = \cfrac{1}{\left(1 + \cfrac{P_{AVE} \times 344400 \times 10^{(1.785 SG)}}{T_f^{3.825}} \right)^2}$$

$$\text{For } P_{AVE} \leq 100, \qquad Z = 1$$

Reference 2

Equation Variables

Unless otherwise noted, the following base units and variable definitions apply to all the methods and equations described in the *Compressibility Methods* section.

P_1 = Pressure at First Conditions, psia
P_2 = Pressure at Second Conditions, psia
P_{AVE} = Average Pressure at Flowing Conditions, psia

SG = Specific Gravity of the Gas Mixture, dimensionless
T_f = Temperature at Flowing Conditions, Rankine
Z = Compressibility Factor, dimensionless

$P_1 = P_{1\_GAUGE} + P_{1\_ATM}$
P_{1_ATM} = Atmospheric Pressure at First Conditions, psia
P_{1_GAUGE} = Gauge Pressure at First Conditions, psig

$P_2 = P_{2\_GAUGE} + P_{2\_ATM}$
P_{2_ATM} = Atmospheric Pressure at Second Conditions, psia
P_{2_GAUGE} = Gauge Pressure at Second Conditions, psig

References

1) American Gas Association, *AGA Report No 8*, Compressibility of Natural Gas and Other Related Hydrocarbon Gases, 1992.
2) Gas Processors Suppliers Association, Engineering Data Book, Eleventh Edition - FPS, 1998.

GASWorkS™ 10.1   *Appendix*

Compressor Power Equations

The equations used by GASWorkS to estimate the compressor power requirements are described as follows.

Adiabatic

The Adiabatic Compressor power equation used by GASWorkS is described as follows...

$$HP \ = \ 0.00007278Q \left(\frac{P_b \times T_f \times Z_f}{T_b \times E_m} \right) \left(\frac{k}{k-1} \right) (R^{\frac{k-1}{k}} - 1)$$

Where,

$$R \ = \ \frac{P_2}{P_1}$$

Reference 2

Empirical

The Empirical Compressor power equation used by GASWorkS is described as follows...

$$HP \ = \ 24Q \left(K_1 \left(\frac{P_2}{P_1} \right)^{K_2} - K_3 \right) \frac{1}{E_m}$$

Where,
$K_1, \ K_2, \ K_3 \ = \ Empirically \ Derived \ Equation \ Constants$

Reference: None - Hand-me-down

Rule-Of-Thumb

The Rule-Of-Thumb Compressor power equation used by GASWorkS is described as follows...

$$HP = 0.000024Q\left(\frac{R}{R+RJ}\right)\left(\frac{5.16 + 124\log(R)}{0.97 - 0.03R}\right)\frac{1}{E_m}$$

Where,

$$R = \frac{P_2}{P_1}$$

$$J = \frac{0.022P_1}{100}$$

Reference 1

Equation Variables

Unless otherwise noted, the following base units and variable definitions apply to all the methods and equations described in the *Compressor Power Equations* section.

E_m = Compressor Efficiency, percent / 100
HP = Power, horsepower
J = As defined in the equation description.
k = Specific Heat Ratio, dimensionless
P_1 = Pipe Inlet (Upstream) Pressure, psia
P_2 = Pipe Outlet (Downstream) Pressure, psia
P_b = Base Pressure, psia
Q = Volumetric Flow Rate at the specified Base Pressure and Temperature, cfh
R = Compression Ratio, dimensionless
T_b = Base Temperature, Rankine
T_f = Average Flowing Temperature, Rankine
Z_f = Compressibility Factor at P_1, dimensionless

$P_1 = P_{1\_GAUGE} + P_{1\_ATM}$
P_{1_ATM} = Atmospheric Pressure at First Conditions, psia
P_{1_GAUGE} = Gauge Pressure at First Conditions, psig

$P_2 = P_{2\_GAUGE} + P_{2\_ATM}$
P_{2_ATM} = Atmospheric Pressure at Second Conditions, psia
P_{2_GAUGE} = Gauge Pressure at Second Conditions, psig

References

1) Gulf Publishing Company, *Pipe Line Rules Of Thumb Handbook*, McAllister, Fourth Edition, 1998.
2) McGraw-Hill Book Company, *De Laval Engineering Handbook*, Third Edition, 1970.

Data List Files

The following table describes the data list files included when GASWorkS is installed. The files may be modified or revised to meet the User's specific needs. All data list files end in a ".lst" extension.

Filename	*Description*
class_location	Contains a list of class locations based on the US DOT 192 code.
class_other	Contains a list of class location exceptions based on the US DOT 192 code.
customer_type	Contains a list of common customer type designations.
efficiency_units	Contains a list of GASWorkS efficiency dimensional units.
flow_units	Contains a list of GASWorkS flow rate dimensional units.
meter_size_type	Contains a list of common meter sizes and types.
pipe_size_group	Contains a list of sizing groups based on material type and other attributes. Contains the "standard" GASWorkS 10 sizing groups.
pipe_size_group_gw90	Contains a list of sizing groups used by GASWorkS 9.0.
pipe_size_group_medm	Contains a list of sizing groups used by the Mechanical Design Module (not implemented) for GASWorkS 10.
pipe_size_group_ukdm	Contains a list of sizing groups used by the UK Design Module for GASWorkS 10.
pipe_size_type	Contains a list of GASWorkS pipe sizes and types.
pipe_specification	Contains a list of common pipe specification types.
pressure_units	Contains a list of GASWorkS pressure dimensional units.
regulator_op_mode	Contains a list of regulator operational types.
regulator_size_type	Contains a list of GASWorkS regulator sizes and types.
uk_joint	Contains a list of joint types used by the UK Design Module for GASWorkS 10.
uk_main_connection	Contains a list of connection types used by the UK Design Module for GASWorkS 10.
uk_main_connection_91	Contains a list of connection types used by the UK Design Module for GASWorkS 9.0.

GASWorkS™ 10.1

Filename	*Description*
uk_main_product_type	Contains a list of product designations for "main" type facilities types used by the UK Design Module for GASWorkS 10.
uk_orientation	Contains a list of orientation designations used by the UK Design Module for GASWorkS 10.
uk_ownership	Contains a list of ownership designations used by the UK Design Module for GASWorkS 10.
uk_package	Contains a list of packaging designations used by the UK Design Module for GASWorkS 10.
uk_reinstatement	Contains a list of reinstatement methods used by the UK Design Module for GASWorkS 10.
uk_service_calc	Contains a list of service line calculation options used by the UK Design Module for GASWorkS 10.
uk_service_connection	Contains a list of service connection types used by the UK Design Module for GASWorkS 10.
uk_service_fitting	Contains a list of service fitting types used by the UK Design Module for GASWorkS 10.
uk_service_governor	Contains a list of service governor types used by the UK Design Module for GASWorkS 10.
uk_service_housing	Contains a list of service housing types used by the UK Design Module for GASWorkS 10.
uk_service_meter	Contains a list of service meter types used by the UK Design Module for GASWorkS 10.
uk_service_product_type	Contains a list of product designations for "service" type facilities used by the UK Design Module for GASWorkS 10.
uk_service_termination	Contains a list of service termination types used by the UK Design Module for GASWorkS 10.
uk_service_type	Contains a list of service types used by the UK Design Module for GASWorkS 10.
uk_size_group	Contains a list of sizing groups based on material type and other attributes common in the UK.
uk_surface_category	Contains a list of surface categories used by the UK Design Module for GASWorkS 10.

Filename	Description
uom_type	Contains a list of various units of measure.
yes_no	Contains a list of yes/no responses.

Database Definition Files

The following table describes the various Attribute Database Definition files included when GASWorkS is installed. The files may be modified or revised to meet the User's needs. All Attribute Database Definition files end in a ".ddf" extension.

Filename	Description
def_catt	Contains data definitions associated with a basic customer attribute file.
def_natt	Contains data definitions associated with a basic node attribute file.
def_patt	Contains data definitions associated with a basic pipe attribute file.
gb_4x_c	Contains definitions similar to the fields contained in the customer portion of the GASBase 4 database. Use this definition if converting an older version of GASWorkS, which has a GASBase database attached.
gb_4x_p	Contains definitions similar to the fields contained in the mains portion of the GASBase 4 database. Use this definition if converting an older version of GASWorkS, which has a GASBase database attached.
ukdm_def_c	Contains customer data definitions used by the UK Design Module for GASWorkS 10.
ukdm_def_p	Contains pipe data definitions used by the UK Design Module for GASWorkS 10.
usdm_def_c	Contains customer data definitions used by the US Design Module (not implemented) for GASWorkS 9.0.
usdm_def_p	Contains pipe data definitions used by the US Design Module (not implemented) for GASWorkS 9.0.

GASWorkS Data Items

GASWorkS uses a propriety database to maintain its files. The database is made up of several data files, with each file containing a set of data fields or items. The basic "public" facing items contained in the customer, node, and pipe data files are described in the following tables. These items can be accessed by the edit functions and will be included in most reports and exported data files.

Customer Data Items		
Name	**Description**	**Data Type**
Hydraulic Data Items		
Adjust Load	Design Factor Adjustment identifier	Boolean
Heating Type	Heating Type used with the "Diversified" node application method	Numeric - Short Integer
Load Status	"On" or "Off" identifier for the Load Status	Boolean
Load Units	Dimensional units for the *Per Unit Load* value	10 Character String
Node Load Application	Node Load Application for the customer's *Per Unit Load*	Numeric - Short Integer
Per Unit Load	*Per Unit Load* value for each customer "unit"	Numeric - Double Precision
Unit Count	Unit Count of the customer feature	Numeric - Short Integer
Branch Type	Branch style of the customer feature - none, trunk, or branch	Numeric - Short Integer
Branch ID	Internal Identification Number of the trunk or branch associated with the customer feature	Numeric - Long Integer
Graphic Data Items		
Service Line Style	Geometric style of the service line - 2-point, or polyline	Numeric - Short Integer
Symbol Color	Color of the Customer Symbol	Numeric - Long Integer
Symbol Style	Style of the Customer Symbol	Numeric - Short Integer
X-Coordinate	X-Coordinate horizontal (east-west position) of the customer's location	Numeric - Double Precision
Y-Coordinate	Y-Coordinate vertical (north-south position) of the customer's location	Numeric - Double Precision

Customer Data Items		
Name	**Description**	**Data Type**
Customer Data Items		
Tap X-Coordinate	X-Coordinate horizontal (east-west position) of the customer's "tap" location	Numeric - Double Precision
Tap Y-Coordinate	Y-Coordinate vertical (north-south position) of the customer's "tap" location	Numeric - Double Precision
Text Display Items		
Display Text Items	"Yes" or "No" display of the Customer Text Items	Boolean
Text Rotation	Rotation Angle of the Customer Text Items	Numeric - Single Precision
Text X	X-Coordinate (east-west position) of the Customer Text Items	Numeric - Double Precision
Text Y	Y-Coordinate (north-south position) of the Customer Text Items	Numeric - Double Precision
Attribute Data Values		
Link ID Number	Link ID (Identification) Number of the Customer	40 Character String
Link To Database	Attribute Database Connection Status	Boolean
Other Values		
Edit Date	Date that a data value associated with the feature was changed.	9 Character String
Internal ID Number	GASWorkS maintained identification number	Numeric - Long Integer
Supply Main Number	Internal Identification Number of the main/pipe that the customer feature is assigned to	Numeric - Long Integer

Node Data Items		
Name	**Description**	**Data Type**
Hydraulic Data Items		
Adjust Base Load	Adjust Base Load by Design Factor identifier	Boolean
Adjust External Load	Adjust External Load by Design Factor identifier	Boolean

Node Data Items		
Name	**Description**	**Data Type**
Base Load	Calculated or entered node's *Base Load* value	Numeric - Single Precision
Base Load Known	"Yes" or "No" identifier for the *Base Load* value	Boolean
Base Load Status	"On" or "Off" identifier for the *Base Load* value	Boolean
Elevation	Height above sea level at the node location	Numeric - Single Precision
External Load	Adjustable *External Load* value if customers are attached to any of the pipe segments associated with the node	Numeric - Single Precision
Fixed External Load	Fixed *External Load* value if customers are attached to any of the pipe segments associated with the node	Numeric - Single Precision
External Load Status	"On" or "Off" identifier for the *External Load* value	Boolean
Hydraulic Status	"On" or "Off" identifier for the *Hydraulic Type*	Boolean
Hydraulic Type	Hydraulic Type of Node	Number - Short Integer
Load Units	Dimensional units for *Base Load* and *External Load* values	10 Character String
Name	Descriptive identifier (name or number) for the node	40 Character String
Pressure	Calculated or entered *Pressure* value	Numeric - Single Precision
Pressure Known	"Yes" or "No" identifier for the *Pressure* value	Boolean
Pressure Units	Dimensional units for the *Pressure* value	10 Character String
Temperature	Calculated or entered average flowing temperature at the node location	Numeric - Single Precision
Temperature Known	"Yes" or "No" identifier for the *Temperature* value	Boolean

Node Data Items		
Name	**Description**	**Data Type**
Gas Properties Values		
Heating Value	Heating Value at the node location	Numeric - Single Precision
Specific Gravity	Specific Gravity at the node location	Numeric - Single Precision
Specific Heat	Specific Heat Ratio at the node location	Numeric - Single Precision
Viscosity	Viscosity at the node location	Numeric - Single Precision
Graphic Data Items		
Allow Intersection Flag	"Yes" or "No" display of the Unbroken Intersection Flag	Boolean
Symbol Color	Displays the color of the Node Symbol	Number - Long Integer
Symbol Style	Style of the Node Symbol	Number - Short Integer
X-Coordinate	X-coordinate (north-south position) of the node location	Numeric - Double Precision
Y- Coordinate	Y-coordinate (east-west position) of the node location	Numeric - Double Precision
Z- Coordinate	Z-coordinate (vertical position) of the Node Location	Numeric - Double Precision
Text Display Items		
Display Text Items	"Yes" or "No" display of the Node Text Items	Boolean
Text Rotation	Rotation Angle of the Node Text Items	Numeric - Single Precision
Text X	X-coordinate (east-west position) of the Node Text Items	Numeric - Double Precision
Text Y	Y-coordinate (north-south position) of the Node Text Items	Numeric - Double Precision

Node Data Items		
Name	**Description**	**Data Type**
Calibration Display Items		
Calibration Type	Type of calibration style	Numeric - Short Integer
Calibration Value	Known value to be checked against	Numeric - Double Precision
Other Values		
Edit Date	Date that a data value associated with the feature was changed.	9 Character String
Internal ID Number	GASWorkS maintained identification number	Numeric - Long Integer

Pipe Data Items		
Name	**Description**	**Data Type**
Hydraulic Data Items		
Diameter Units	Dimensional units for the *Size/Type Code* value for *Pipe* Hydraulic Types	10 Character String
Equation	Equation defining the pipe flow-pressure drop relationship	Numeric - Short Integer
Flow Rate Units	Dimensional units for the *Flow Rate* value	10 Character String
Hydraulic Efficiency	• Hydraulic Efficiency for *Pipe* and *Fitting* Hydraulic Types • Mechanical Efficiency for *Compressor* Hydraulic Types	Numeric - Single Precision
Hydraulic Length	• Hydraulic Length for *Pipe* Hydraulic Types • Set Pressure for *Compressor* and *Regulator* Hydraulic Types • Percent Open value for *Valve* Hydraulic Types • Number of wells for *Well* Hydraulic Type	Numeric - Single Precision
Hydraulic Type	Hydraulic Element Type identifier	Numeric - Short Integer
Length Units	Dimensional units for the *Hydraulic Length* item	10 Character String
Pressure Drop Units	Dimensional units for the calculated *Pressure Drop* value	10 Character String

Pipe Data Items		
Name	**Description**	**Data Type**
Roughness	Inside Pipe Wall Roughness value for *Pipe* Hydraulic Types	Numeric - Single Precision
Size/Type Code	• Size and Type description for the *Hydraulic Type* • *Inside Diameter* value for *Pipe* Hydraulic Types if not selected from the associated list	20 Character String
Status	"On" or "Off" indicator for the status of the *Hydraulic Type*	Boolean
Pipe Sizing Items		
Allow Sizing	Allow Sizing indicator for *Pipe* Hydraulic Types	Boolean
Sizing Group	Sizing Group for *Pipe* Hydraulic Types	Numeric - Short Integer
Graphic Data Items		
Pipe Line Color	Display color of the pipe line	Number - Long Integer
Pipe Line Style	Style of the pipe line	Numeric - Short Integer
Pipe Line Thickness	Thickness (width) of the pipe line	Numeric - Short Integer
Pipe Line Type	Geometric style of the pipe line - 2-point, polyline, arc	Numeric - Short Integer
Text Display Items		
Display Text Items	"Yes" or "No" display of the Pipe Text Items	Boolean
Text Rotation	Rotation Angle of the Pipe Text Items	Numeric - Single Precision
Text X	X-coordinate (east-west position) of the Pipe Text Items	Numeric - Double Precision
Text Y	Y-coordinate (north-south position) of the Pipe Text Items	Numeric - Double Precision
Miscellaneous Data Values		
Facility Type	Facility Type assignment	Boolean

Pipe Data Items		
Name	**Description**	**Data Type**
Group ID Number	Identification number of the group the pipe assigned to (if any)	Numeric - Long Integer
Group Type	Type of group the pipe was assigned to (if any)	Numeric - Short Integer
Attribute Data Values		
Link ID Number	Link ID (Identification) Number of the pipe segment.	20 Character String
Link To Database	Attribute Database Connection status	Boolean
Other Values		
Edit Date	Date that a data value associated with the feature was changed.	9 Character String
Internal ID Number	GASWorkS maintained identification number	Numeric - Long Integer
MAOP	Maximum Allowable Operating Pressure of the pipe	Numeric - Single Precision
Attached Fittings	Number of attached fittings	Byte

Data Type Descriptions	
Type	**Description**
XX String	Represents a group of alphabetic (A - Z) characters, numeric (0 - 10) characters, and some special characters up to XX.
Boolean	Represents an either/or type value, such as "On" or "Off" or "Yes" or "No." Stored as a Short Integer.
Byte	Represent a numeric value represented by a single byte.
Numeric - Double Precision	Represents a decimal number. See the *GASWorkS Limitations* topic in the *Miscellaneous* section for additional information.
Numeric - Single Precision	Represents a decimal number. See the *GASWorkS Limitations* topic in the *Miscellaneous* section for additional information.
Numeric - Long Integer	Represents a non-decimal (whole) number. See the *GASWorkS Limitations* topic in the *Miscellaneous* section for additional information.

Data Type Descriptions	
Type	**Description**
Numeric - Short Integer	Represents a non-decimal (whole) number. See the *GASWorkS Limitations* topic in the *Miscellaneous* section for additional information.

GASWorkS Files

The following table describes the various files used by GASWorkS. Depending on the nature and features included in a specific model and the routines used, some of these files may not always be present.

Category	*Filename*	*Description*
Back-Up Files	.bak	Back-Up Data file
Model Data Files	.cdt	Customer Model Data file
	.hdr	Header Data file
	.ndt	Node Data file
	.pdt	Pipe Data file
Non-Model Data Files	.cdb	Linked Customer Attribute Data file
	.ftg	Attached Fitting information
	.gtx	User Text file
	.nts	Model Notes file
	.pdb	Linked Pipe (Main) Attribute Data file
	.prp	Gas Properties file
	.ugd	User Graphics file
	.vtx	Vertex Location file
Program Files	gw10.ini	Contains the screen locations, Path Settings, Preference Settings
	gw10calc.dll	Contains the various support calculations used by GASWorkS
	gw10gdi.dll	Contains the GASWorkS Graphic Data Interface functions
	gw10lic.dll	Contains the GASWorkS license information
	GW100.exe	The GASWorkS Program file
	gws.dll	Contains the GASWorkS solution functions
Property Tables	.cpf	Compressor Property Table
	.fpf	Fitting Property Table
	.lut	Look-Up Table - used by older versions of GASWorkS
	.ppf	Pipe Property Table

Category	*Filename*	*Description*
	.rpf	Regulator Property Table
	.rtb	Regulator Capacity Table
	.vpf	Valve Property Table
	.wpf	Well Property Table
Screen Initialization Files	.ini	Contains information about the previous screen location and settings
Support Files	.att	Attribute Data Settings
	.bmp	Contains a saved GDI display image
	.bom	Contains Bill-Of-Materials content
	.bom.***	Contains Bill-Of-Materials content for UK Design Module
	.chk	Data Check Error file
	.csv	CSV file containing User Table or other User created data
	.cxi	Saved CSV Import Settings file
	.dck	Data Check Settings file
	.ddf	Saved Attribute Database Definition
	.dfs	Saved Default Data Values Settings file
	.dft	Default Data Values Settings file
	.dxe	Saved DXF Export Settings file
	.dxi	Saved DXF Import Settings file
	.err, .etp	Node Error file
	.exc	Solution Exceptions file
	.fac	Facility List and Settings file
	.fck	Temporary file used by some File Management routines
	.gpj	GASWorkS Project file. Saved settings from the Project Specifications screen
	.grf	Graphic Settings file
	.img	Background Image Settings file

Category	*Filename*	*Description*
	.kme	Saved KML Export Settings file
	.kmi	Saved KML Import Settings file
	.lst	Data List file
	.nxi	Saved Node Import Settings file
	.pfs	Preference Settings file
	.prj	Project Specification file
	.pvw	Model Graphic Preview file
	.qry	Saved Query Specifications file
	.qwk	Quick Size Specifications files
	.scl	Saved customer SQL Specifications
	.snl	Saved node SQL Specifications
	.spl	Saved pipe SQL Specifications
	.slg	Solution Log file
	.sum	System Summary Report Settings file
	.sxi	Saved SHP Import Settings file
	.tbd	Saved Title Block Definition
	.tmp	Used by various routines to store temporary data
	.trc	Trace Results file
	.txt	Text file containing export or reported data or information
	.unq.txt	Unassigned Customer file. Created during execution of the Update External Loads routine.
	.vew	Saved Graphic View file
	.wkt	Coordinate Project file in "well-known text" format
	.zip	Compressed file containing bundled model files

GASWorkS™ 10.1   *Appendix*

Network Primer

The examples used in this discussion often refer to distribution-type piping systems. However, the general concepts described in this section apply to all piping systems. Hopefully, the User can make the analogies between the cited examples and their particular application. Additional notes are provided at the end of this section for working with other types of systems.

About Network Modeling

The following provides a brief description of network modeling, the benefits derived from modeling, and some general information concerning gas networks. It assumes you have a working knowledge of the design, analysis, and operation of gas systems and the terms normally associated with the field. If you are not completely familiar with all of the terms or concepts you encounter, you may want to refer to an industry handbook or guide.

For our discussion here, a model is a mathematical representation of a gas system. It is created by defining a group of interconnected nodes and pipes. A node represents the beginning or ending point of a pipe. Nodes are connected by pipes. Load is brought into the system or taken out of the system through nodes. Nodes not only represent pipe connections but can also represent customer connections and supply points.

Pipes carry gas between the nodes. The sum of the pipe flows going into a specific node must equal the sum of the flow leaving the node through the connected pipes plus the flow leaving the system through the node. Similarly, the sum of the flow coming into the system through a node must equal the sum of the flow leaving the node

through the connected pipes. In steady-state modeling, neither nodes nor pipes can store flow.
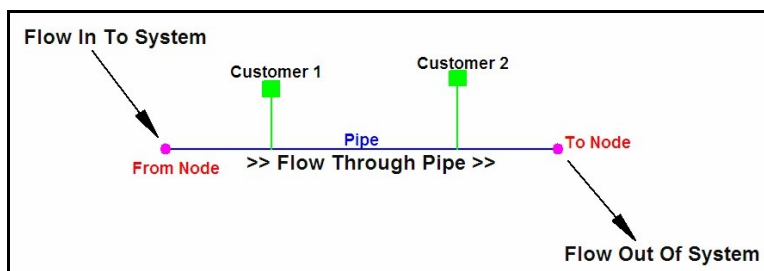
A further characteristic of steady-state network behavior requires the sum of the pressure drops from one point to another along any path to be the same. In other words, a node point can possess only one pressure value.

Flow in a pipe moves from the higher-pressure end to the lower-pressure end. That is, flow always moves toward the lower pressure node. Like a ball rolling downhill, gas flows from high to low. Flow is created by a difference in pressure, and consequently, flow causes frictional pressure losses. Generally, as the flow increases, the frictional losses also increase. The greater the pressure difference, the greater the flow rate. The greater the flow rate, the greater the frictional pressure losses.

Because of the nature of the equations that describe the pressure-flow relationship in a gas network, the pressure and flow values in a gas system cannot always be directly calculated. The values must usually be determined by recursive calculation. To control this recursive process, an attempt is made to balance the flow into and out of all of the nodes in the model. When the imbalance of these flows is within a specified tolerance, the model is said to be "balanced." However, it is unusual for the solution of a model to result in an imbalance exactly equal to zero (0) at all nodes. To account for this behavior, a Convergence Tolerance is used to check the balance condition.

The Convergence Tolerance represents the amount of imbalance acceptable for a particular analysis. The Convergence Tolerance is analysis-specific and should be based on certain peculiarities of the analyzed model. When establishing the Convergence Tolerance, consideration should be given to several factors:

- The coarseness of the model data, that is, how accurately the model data is known.

- The magnitude of the total flow into the system being modeled.

- And the intended use of the results from the analysis.

An acceptable Convergence Tolerance is usually less than one percent of the total anticipated flow into the modeled system.

A "flow equation" is required to describe the relationship between the flow in the pipes and pressures at the nodes. Most flow equations require the definition of certain parameters describing the gas in the system and the physical properties of the pipes. A friction factor may be inherently assumed by the equation or calculated depending on the equation format. Many equations are available for modeling the conditions normally encountered in gas distribution, transmission, and gathering systems. The challenge is selecting the one that best suits your specific application.

A model is only as good as the data used to build it. Analysis based on incorrect values will yield incorrect results. The analysis results should be evaluated using the proper weight. In most gas distribution models, customer usage and load distribution will be the most critical parameters in the analysis, with the physical system configuration being the second most critical group of data.

The Benefits of Modeling

There are several benefits derived from creating and analyzing computer models of a gas system. Some of these are:

- The ability to anticipate operating conditions within the system.

- The ability to identify problem areas and troubleshoot problems before they become serious.

- The ability to efficiently size new and replacement pipe segments to the system.

● The ability to estimate the impact on the system by adding new customers or supplies.

● The ability to analyze "what if" scenarios without physically manipulating the gas system or operating settings.

Modeling a system helps prevent potentially unnecessary expense by allowing design and operating ideas to be tested using the model, as opposed to the potentially costly method of testing the ideas in the field. Computer modeling allows alternatives to be tested and refined before implementing them. It can help avoid the construction of improperly sized facilities and avoid making modifications to the system or operations that could adversely affect its serviceability.

About Networks

A "network" is any system of interconnected or interrelated components. In a network, each component, to some extent, affects every other component. A gas system may be considered as a network. In a gas system, the components of the network are the pipes, valves, fittings, connections, and regulators that make up the physical configuration of the system - and the customers attached to the system.

As an example of how individual components of a gas network might affect the other components in the network, consider the following:

● The outlet pressure of a regulator supplying the system is reduced. To a certain extent, the pressure in the entire system might be changed - the pressures upstream of the regulator will likely increase, and pressures downstream of the regulator will decrease. If more than one regulator supplies the portion downstream of the regulator, flows will be redistributed.

● An additional regulator is added to the system, and the flow from the other regulators will be changed. The flow through the pipes will be redistributed to accommodate the change in flow from the regulators.

● A valve in the system is closed. Flow in the system must be redistributed to accommodate the changes caused by the valve closure. The flow and pressure in all pipes will be changed to a certain extent. The flow will decrease in the mains to which the valve was attached and increase in the mains, which might provide flow around the valve. When flows change in the system, so do the pressures in the system.

● A large load is connected to the system. The flow in the portions of the system supplying that load will be increased. The increase in flow will affect the pressures in the system and the flow required of the supplying regulators.

These few examples demonstrate the fundamental concept of a network - that within a network, every member is essentially connected to and affected by every other member in the network. A network may be considered a large machine with each component working together with the other components to make the machine operate.

Flow In Gas Networks

If you refer to the previous definition of a gas network, you will recall that the network consisted of pipes, fittings, regulators, and customers. Customer usage is normally what causes flow in a gas system. When it is cold, the customers' furnaces operate quite frequently, creating a demand for the gas system. The flow in the gas system might be very high to satisfy this demand. When it is warm, generally only pilot lights are operating, and their demand is quite low. Consequently, the flow in the system is very low at that time. Let's look at this phenomenon closer. What happens in this process that causes the gas to flow through the system?

Consider a system that is comprised of a single customer, a single segment of the main, and a single supplying regulator. When the customer's home gets cold, the thermostat in the house reaches a point where it turns on the furnace. When the furnace is turned on, the gas valve opens, allowing gas to enter the furnace. In a conventional furnace, the absolute pressure is the same as the atmospheric pressure. That is, the gas is released into the atmosphere. This is true because the combustion chamber of the furnace is open to the atmosphere through the flue and the inlet air valves. The gauge pressure then, at the end of this network of regulator, main, and customer, is zero (0). In other words, if a pressure gauge were held in the combustion chamber of the furnace, the gauge would read zero (0).

If the network had no supply source and the furnace continued to operate, the pressure in the main supplying the customer would decrease over time. Eventually, the pressure in the main would be reduced to the same pressure as the combustion chamber of the furnace, zero (0).

This system, however, has a source of supply, the regulator. Initially, when the furnace is off, the regulator is closed. There is no gas coming into the system. As the furnace continues to use the gas from the main, the pressure in the main begins to decrease. The regulator will eventually sense this decrease and open slightly to attempt to maintain the pressure in the main. The regulator will continue to open as required to satisfy the demand imposed by the consumption of the furnace. In theory, the amount of gas supplied by the regulator will eventually equal the amount of gas consumed by the furnace. When this happens, the conditions in the system are said to be in a steady state.

In summary, the demand for gas by the furnace in the customer's home causes gas to flow through the system to satisfy that demand. When the supply through the system is equivalent to the demand on the system, the conditions in the system are said to be steady state.

Let's look at the conditions occurring in the system during steady-state flow. The pressure at the end of the system (the customer's furnace) is zero (0). Let's assume the pressure at the regulator supplying the system is 20 psig. The pressure in the main that connects the customer to the regulator, therefore, varies from 20 at the regulator to 0 psig at the furnace. This difference in pressure causes gas to flow through the main line. The amount of gas flowing through the main exactly equals the amount of gas entering the system at the regulator and the amount of gas leaving the system through the furnace. The flows and pressures are unchanging (steady); therefore the system conditions are in a steady state.

When the furnace turns off, the customer end of the system will no longer be open to the atmosphere. However, for a time, a pressure difference will exist in the system, and the gas will continue to enter the system through the regulator, but no gas will leave through the furnace. During this time, the flow in the main equals the flow entering through the regulator but does not equal the flow leaving through the furnace (which is zero (0)). Further, as the gas enters the system, the pressure at the furnace end will continue to increase. The flow into the system does not equal the flow leaving the system, and the pressures change. These conditions are referred to as unsteady or transient. Eventually, the pressure in the system will equalize (no pressure difference), and the flow will cease. When this occurs, the system will again be in a steady state.

When gas is flowing, frictional pressure losses occur along the pipe segments. The magnitude of the frictional pressure losses is affected by several factors. A decrease in pipe size, restrictions in the system such as valves or debris, and increased roughness of the inside pipe wall will cause the magnitude of the frictional losses to increase. The composition and physical properties of the gas also affect the magnitude of the frictional losses. For a given flow rate, an increase in specific gravity, the flowing temperature, or the viscosity generally causes the magnitude of the frictional losses to increase.

Determining Model Loads

As mentioned in the previous section, the demand or load placed on a gas system is a function of its customers' usage characteristics. To model a gas system, an estimate of the customer loads must be determined. This is probably the most critical piece of data that will be used in the model. Unfortunately, this is generally the least known piece of the required data.

Several methods can be used for determining the value of these loads. A few of them are described in the following sections.

The first method attempts to determine actual usage values (generally peak-hour usage values). The other attempts to determine the total connected load (TCL) for each customer and then applies a usage factor (Design Factor) to the TCL. Either method is acceptable if applied correctly and judiciously.

Billing File Method

With this method, the "per unit load" value for each customer feature is assigned to its associated value from a selected monthly billing file. The theory behind this method is that the monthly usage for each customer is proportional to their hourly usage of the system total for a period occurring during the associated billing month. The general steps required to determine the customers' hourly loads by this method are:

● Create customer features in the model manually or by importing or merging customer data from a data file. Assign a unique number to the Link ID value of each customer. The value should correspond to a similar value in the billing file to be used.

● Obtain a copy of the billing file for one or more months of usage. The file should include, at a minimum, the unique number assigned to the customer features in the model and the monthly usage for the month(s) corresponding to the peak hour value to be used. The monthly usage should be listed separately for each customer for each month contained in the file. The file may contain additional information, such as address or class of service.

● Convert the billing file to an "attribute" format supported by GASWorkS. Attach the converted file to the model using the *Attribute File* item from the *Customer Data* submenu of the *Utilities* menu. Assign the data fields and dimensional units as appropriate. Ensure that the Connection Status is set to "on."

● Use the *Update External Loads* item from the *Customer Data* submenu of the *Utilities* menu to assign the usage values in the Billing File to the associated customer features in the model. Ensure that the Link To Database value is set to "Yes." This step will assign the values in the billing file to the matching customer features in the model. If the dimensional units are set to a "pre-month" style unit, the model values will represent average hourly values (based on per-month usage) when used in the solution and using a Design Factor value of 1.00.

● Obtain the total hourly system flow values for the period you wish to model, for example, for a peak-hour period during cold weather. Open the Solution Data screen using one of the Solve commands. For the Design Factor value, either enter or calculate and apply a value that causes the total system flow to match the desired hourly flow value. The corresponding Design Factor value may be calculated by selecting the *Calculate* command button adjacent to the *Design Factor* data field.

● The resulting total system flow may be calculated using the Supply Nodes Summary or System Summary options with the System Summary Report routine in the Report menu.

Proration Method

With this method, individual customer load values are determined by prorating their monthly usage values concerning the total usage for a given period (usually a peak month). This method produces results similar to those of the Billing File method. However, it involves more steps and is more difficult to explain. The steps required to determine the customers' peak-hour loads by this method are:

● Determine the historical peak-hour flow rate for the entire system being modeled. Usually, this occurs coincidently with peak-day events, but it does not always happen. If only daily or monthly demands are available, an appropriate factor may be applied to estimate the hourly demand. Generally, peak-day to peak-hour conversion factors range from 1/16 to 1/20.

● Determine the usage (usually monthly values) of each customer during the peak demand period. The data should include a customer identifier (address, facility ID, premise ID, etc.) and the usage or billing values over the month(s) representing the peak period. Calculate the average usage for each customer and the portion of the total load each customer uses.

● To determine each customer's portion of the total load, sum the usage values for all customers. Then, divide each customer's usage by the total usage; the result represents the customer's portion of the total usage.

Note - If the above calculations are performed using a spreadsheet, it may be possible to link the resulting "database" to GASWorkS and use the external database features to automatically update the model loads.

● Finally, prorate the total peak-hour value determined in Step 1 for each customer. To prorate the flow, multiply the total peak-hour value by the prorated usage (determined in Step 2) for each customer. This result will approximate the peak-hour usage of each customer. When properly applied to the model, these loads may be used to estimate the peak-hour performance of the system being modeled.

● This method can also be used for off-peak periods using an appropriate total hourly value.

Note - When using this method, generally, the customer loads are input as the usage factors computed in Step 2, and the Design Factor is set to the peak-hour value computed in Step 1 or another hourly total as suggested in Step 3.

Total Connected Load

With this method, the total rating of the appliances and equipment at each customer location is determined and applied as the load value. The total rating is sometimes called the Total Connected Load or TCL. The following discusses the use of this method:

● There are several methods to determine the customers' total connected load. The best method is to survey each customer location to determine the input rating of each appliance. However, this is not always practical. A compromise method is to visit only the commercial and industrial type customers and a portion of the residential customers. Collect the individual loads for each commercial customer visited, then select a random sampling of residential customers. Collect their loads and calculate an "average" residential load based on the collected data. Apply this average to the remaining residential customers.

● If there is a great dispersion of residential construction types and ages, group the residential customers into classes and calculate an average for each class. For example, homes built before 1940, homes built between 1940 and 1950, or homes built in the 1960s might represent classes of residential customers. Another good way to group residential customers is by subdivision or development. Often, homes built by a particular developer will have appliances of similar size and construction types. Of course, if resources permit, the most accurate way to determine the loads of your customers is to visit each home and collect the data.

● Once estimated or collected, the customer loads may be applied to the model. However, the loads must be adjusted during the analysis based on a usage factor. A usage factor may be estimated by dividing the peak-hour flow rate by the sum of the customer loads. This factor can be applied during the solution by setting the Design Factor equal to the usage factor. The value of the Design Factor ranges greatly depending on the number of customers being served and the climate in the area where the system is located.

Regardless of the method used to determine the customer loads, it is time-consuming. Avoid the temptation to be lax in your efforts to complete this task. Remember, the model will be used to make very expensive and critical decisions; therefore, the model must be as accurate as practical - the customer loads are one of the most critical pieces of the model's data. You will find that a little extra effort spent accurately determining the customer loads now will be resources well spent in the future.

Creating Gas Network Models

To accurately model a gas system, it is necessary to determine two fundamental types of data. The first is to determine the physical configuration of the system. That is, it is necessary to determine the size and length of each main segment, how and where each main is connected to the other mains, the location of any flow or pressure-controlling devices (regulators, compressors, or closed valves), and the location of the supply and demand points for the system.

Usually, the system configuration is determined from system maps or system records. If the records are incomplete or inaccurate, it is often helpful to interview the field personnel responsible for maintaining and operating the system. Their insight may be quite useful in determining the actual configuration of the system.

Secondly, the system's load (demands and supplies) must be determined. Use one of the previously described methods to estimate the loads associated with the modeled system.

Once all of the data is collected, it may be directly used to create the model, or it may be used to create an intermediate schematic of the system. A schematic is a depiction or drawing showing the configuration of the piping system. The schematic might be an actual paper drawing, or it might be created using the GASWorkS Graphic Data Interface or a drawing created using a CAD or GIS system. In the schematic, show each main and its connection to each other. The schematic does not have to be scaled (however, it is best if it is), but it should accurately reflect how the system is configured.

The details required for a model vary depending on its intended use, complexity, and the data available to create the model. Rarely is a node placed at every customer location or valve location, or every pipe size or type change included in a model. The model configuration is defined by its node placement. When constructing the model and where to place nodes, consider the following placement criteria. Generally, a node should be "placed" at the following locations:

- At each location where two or more mains intersect and are connected.

- At each location where a main changes size, wall thickness, or material.

- At each supply point in the system.

- At each main termination.

- At large demand locations. Normally, customer loads may be generalized at the piping connection nodes. However, additional nodes should be placed where a large customer connects to the system or locations where the distance from the closest piping node and the point at which the customer's physical connection to the system is substantial.

To compensate for fluctuations in atmospheric pressure, the elevation of each node will need to be determined. If the total variation in elevation across the system is less than 1000 feet, determine the average elevation of the modeled system and apply the average value to each node.

If the schematic is created using a paper or CAD drawing, include the size and length of the main segments on the schematic and any significant node Names. If the schematic was not created using the GASWorkS GDI tools, the data will now need to be either manually entered or imported into GASWorkS using the appropriate method.

Once the system configuration has been entered and checked, the next task is to assign the customer loads. There are several methods used for assigning customer loads. The methods that we most commonly recommend are described as follows.

- Allocate loads for customers dispersed along a pipe segment (not using customer data features):

    - Sum the load along each main segment connected to an individual node.

    - Divide this sum by two.

    - Apply the resulting value to the pipe's From Node and To Node.

This method works well with looped, "two-way" flow configurations. When working with a "one-way" flow (skeleton-type system), consider applying all of the load along a segment to the downstream node of that segment. This will result in conservative pressure drop values. To be slightly less conservative, apply 2/3 of the segment's load to the downstream node and 1/3 to the upstream node.

- Automatically allocate loads along a main using customer data features:

    - Place and attach customer features along the main segments as appropriate.

    - Enter each customer's specific load or a "generic" load for the associated customer type.

    - Unless the customer represents a very large load, apply the load to "Both Nodes." For large loads, apply the load to the appropriate "From Node" or "To Node."

Regardless of which assignment method is employed, discretion should always be used when applying the loads. Large loads may warrant that a specific node be placed at their connection point. GASWorkS convention requires that a demand (a flow leaving the system) be indicated by a negative value and a supply (a flow entering the system) be indicated by a positive value.

Now that the physical configuration of the distribution system has been defined and the customer demands have been determined and allocated, information about the gas flowing through the system and a few other parameters need to be determined. These items are:

- A gas composition analysis for the gas taken around the modeled period. Or, if that is not available, the Specific Gravity of the gas compared to dry air and the absolute viscosity of the gas.

- The average flowing temperature of the gas during the season to be modeled.

- The base pressure and temperature used to state volumes. These values represent the basis for measuring the gas - they are usually the same as the billing pressure and temperature.

GASWorkS™ 10.1 *Appendix*

The gas analysis (or properties) can usually be obtained from the gas supplier. If an analysis is available, use the Calculate Gas Properties routine to calculate the required gas property values.

To complete the "mathematical model," a correct flow equation needs to be determined. The operating pressure and system characteristics must be considered when selecting the appropriate flow equation. There are a variety of equations available to model gas pressures and flows. You will need to select the equation that best suits your specific application. Refer to the *Equation Application Reference* topic in the *Appendix* section for discussion and suggestions on selecting an appropriate flow equation.

Finally, the known operating conditions of the system need to be input. In general, GASWorkS requires that either the pressure or load be "known" at each node point - but not both. There are exceptions to this rule for regulators and compressors. Generally, the pressure value will be set as "known" at each node point representing a supply, and the load value will be set as "known" at nodes representing a demand - the flow or pressure is set to "unknown" as appropriate. Nodes that represent a connection with no flow entering or leaving a system (node at a pipe size change or tee) are generally set so that the load is "known," zero (0), and the pressure is set to "unknown." All unknown values are computed during the analysis process.

Sometimes, especially at supply points, both the flow and pressure values might be "known" - the normal tendency would be to set both these values as "known" - whereas this is possible to do, it generally results in an "over-constrained" model that will not solve or will yield unusual results. In these cases, it is generally best to set one of the values as "known" (usually the pressure value) and let GASWorkS solve for the other. This un-constrains the model, lets it solve, and also provides a check for its accuracy.

The culmination of the above data represents the basis for the "model." This data is used to develop the mathematical model used by GASWorkS. As mentioned, the data may be directly entered into GASWorkS or imported using one of its Import routines or a combination of these methods. Once entered, the model can be manipulated and revised to correct, test, and verify the model.

Whenever possible, when working with an existing system, the solution results should be checked against actual field data - the model should be "calibrated" to accurately reflect field conditions. This may be done by comparing GASWorkS-generated pressure and flow values with field-collected data. Always check at several locations in both the system and the model. A model can always be forced to match a single specific value at a single specific point. However, the proof of a model is in the ability to match values at several locations in the modeled system. To calibrate a model, model results may be forced to match field conditions by adjusting the pipe efficiencies as necessary to make a match.

Working With Gathering Systems

When working with gathering systems, a few basic issues should be considered. A gathering system is essentially a backward distribution system. It generally has one "demand" point and many supply points, as opposed to a basic distribution system with one supply and many demands. The supply point on a distribution system is usually equipped with a regulator, and the demand point for a gathering system is generally equipped with a compressor. The construction of the basic piping model is essentially the same as that of a distribution system - the application of the "loads" and settings vary, however.

For existing systems, there is generally quite a bit of information known about the systems operation - flow and pressure rates at every well and possibly a flow rate and suction or discharge pressure at the compressor (demand point) are known. GASWorkS generally requires that either the pressure or load value be set as "unknown" at each node. When working with a gathering system where much data is known, it may be challenging to decide which values should be fixed ("known") and which should be allowed to be calculated ("unknown"). In a gathering system, there are two ways to address this issue - set the well pressures as "known" and let the flow rates be calculated, or set the well flows as "known" and let the pressures be calculated. How the "knowns" and "unknowns" are set up at the well nodes depends on what information is known and how the wells are operated (whether flow or pressure control is present).

When the well pressures are fixed ("known"), the flow rates are calculated - the results from an analysis setup in this manner will provide the estimated flow rates from the wells while holding the well outlet pressures at a fixed value. If a new well is added to a model using this configuration, generally, the flow rates in the other wells will be reduced because of the increased downstream pressure attributable to the new well. This type of analysis would provide an estimate of how much the existing well production might be reduced by the addition of the new well.

When the well flows are fixed ("known"), the pressure values are calculated - the results from an analysis setup in this manner will provide the estimated pressures at the wells while holding the well flow rates fixed. If a new well is added to a model using this configuration, the pressures at the other wells will generally be increased because of the increased downstream pressure attributable to the new well. This type of analysis would estimate how much the existing well pressures would need to be increased to maintain their previous production.

GASWorkS allows wells to be modeled by two different methods. The most often used method uses a single node to represent the well location. With this method, the node basically represents the wellhead, or if processing or control equipment is present, the downstream connection to the equipment, or the point of flow and pressure measurement. The other method models the Well as a *Hydraulic Type* feature. This method requires that the operating characteristics of the well be known or estimated. These factors are usually determined from a "multi-point" flow test - which is not often available or outdated.

At the compressor end of the system, the compressor may be modeled as a *Hydraulic Type* feature, or the system model can end at the suction side of the compressor. When a compressor-type feature is present, generally, the load on the inlet (suction) side of the compressor is set to "known" and zero (0). If the well pressures are set as "known" and the loads as "unknown," the inlet (suction) pressure is generally set as "unknown." If the well flows are set as "known" and the pressures as "unknown," the inlet (suction) pressure must be set as "known." The downstream (discharge) pressure is generally set as "known." When entering a compressor-type feature, the From Node must represent the inlet (suction) side of the compressor, and the To Node must represent the outlet (discharge) pressure side of the compressor.

Often, the reason to model a gathering system is to identify "bottlenecks" or restrictive pipe segments. The results produced by GASWorkS may be used to identify these locations by two methods. The first and most obvious method is reviewing the reported segment pressure drops. The other is to attempt to calibrate the model by adjusting the pipe efficiencies to force the model results to match the field recorded values - usually pressure values. Pipes that require low-efficiency values (less than 80%) indicate problem areas, usually resulting from liquid buildup.

Be careful when comparing pressure values between field measurement equipment and GASWorkS. Often, flow meter or flow computer data are reported in absolute pressure values, while all node pressure values reported in GASWorkS are gauge values.

The Weymouth flow equation has been traditionally used to design gathering systems; however, due to its conservative nature, it does not tend to match field results very well. Depending on pipe sizes and operating pressures, the IGT-Improved flow equation tends to match field results with good repeatability.

GASWorkS provides three compressor equations:

- Empirical - Requires empirical data from a compressor operating curve.
- Adiabatic - Uses theoretic thermodynamic and compression characteristics.
- Rule-Of-Thumb - Uses empirical and theoretic data to approximate the compressor characteristics.

The Adiabatic and Rule-Of-Thumb equations provide good estimates of compressor performance values when used judiciously. GASWorkS models all compressors as a "black box" and does not use any of the compressor characteristics in the hydraulic analysis.

Working With Plant (Fuel Piping) Systems

When working with plant or fuel piping systems (systems downstream of the gas meter), a few basic issues should be considered. These systems represent a small (or not so small) distribution system. Generally, these systems have one supply source and do not contain internal looping (interconnected supplies). The construction of the basic piping model is essentially the same as that of a distribution system - the application of the "loads" and settings vary, however.

Instead of "customers," these systems generally connect directly to the demand (equipment). For this reason, a node is generally placed at each equipment connection. The rated load of the equipment is generally "known," and the supply pressure is nearly always "known."

When modeling a distribution system, it is seldom considered that all of the demands will be in service at the same time. However, with fuel piping systems, depending on the number and nature of the equipment that is attached, it is not unusual for all of the demands supplied by a fuel piping system to be on at the same time. The fewer the number of appliances or equipment supplied by the system, the greater the chance that all of the devices will be on at the same point in time.

When setting up a model of a fuel piping system, generally, a node is placed at each appliance or equipment connection. The load is usually set as "known," and the pressure is set as "known" at these locations. A node is placed at the supply point, and the pressure is set to "known" and equal to the delivery value. The appliance nodes generally represent the connection to the appliance control valve or regulator. The supply node generally represents the outlet of the meter or the service regulator, whichever is further downstream.

GASWorkS provides several pipe flow equations specifically applicable to modeling fuel piping systems. These equations are recognized by most mechanical and fuel gas codes. The Oliphant equation is often cited as applicable to "plant" piping operating at pressures ranging from vacuum to 100 psig.

Working With Transmission Systems

When working with transmission systems, a few basic issues should be considered. For this discussion, a transmission system is considered the portion of the overall gas system that carries gas from a gathering system or group of systems to a processing plant, distribution system, or another transmission system - or from another transmission line or plant to a distribution system. It characteristically contains larger diameter pipe, operating at comparably high-pressure, has long distances between connections or interconnections, and may contain one or more compressor stations. The construction of the basic piping model is essentially the same as any system's - the application of the "loads" and settings vary, however.

The steady-state nature of the GASWorkS Solution routine is suitable for modeling a transmission system with those characteristics - steady, non-transitory operating conditions. GASWorkS is not suitable for modeling transient conditions.

Transmission systems generally have a few sources of supply and a few demand points, often with compressors between the two. A supply point is generally modeled as a node point with a "known" pressure and an "unknown" load. Demand points are generally modeled as node points with a "known" load value, and pressure values are set "unknown."

When including compressors, generally, the load on the inlet (suction) side and outlet (discharge) side of the compressor is set to "known" and zero (0) - and the outlet (discharge) pressure is set to "unknown." When entering a compressor-type feature, the From Node must represent the inlet (suction) side of the compressor, and the To Node must represent the outlet (discharge) side of the compressor. Pressure must be known between each model section, separated by a compressor.

GASWorkS provides three compressor equations:

- Empirical - Requires empirical data from a compressor operating curve.
- Adiabatic - Uses theoretic thermodynamic and compression characteristics.
- Rule-Of-Thumb - Uses empirical and theoretic data to approximate the compressor characteristics.

The Adiabatic and Rule-Of-Thumb equations provide good estimates of compressor performance values when used judiciously. GASWorkS models all compressors as a "black box" and does not use any compressor characteristics in the hydraulic analysis.

Various flow equations are available for modeling transmission-style systems. The operating pressure and system characteristics must be considered when selecting the appropriate flow equation. There are a variety of equations available to model gas pressures and flows. You will need to select the equation that best suits your specific application. Refer to the *Equation Application Reference* topic in the *Appendix* section for discussion and suggestions on selecting an appropriate flow equation.

Often, transmission systems operate at pressure values sufficient to include the effect of compressibility in the pipe flow calculations. GASWorkS provides several methods for including the effect of compressibility. Several of the equations require that a gas composition be present to support the calculation. These values are saved in a gas properties file. Use the *Calculate Gas Properties* item in the *Analysis* list to create and save a gas properties file. The file must be named the same as the model it is to be used with. Currently, the most accurate method of representing compressibility supported by GASWorkS is the AGA8-92-Detailed method. The worst method is the GPSA method. Using the GPSA method will yield unconservative results at pressure ranges over about 400 psig. Ignoring compressibility all together will yield conservative results. Sometimes, compressibility will be ignored during design analysis to yield conservative results but will be included when trying to match existing conditions. The use of compressibility will slow down the Solution routine and will increase the number of iterations required to reach the solution.

Many pipe flow equations ignore or assume the internal pipe wall roughness value. However, the pipe wall roughness is used in some transmission-style equations. Recommended values for pipe wall roughness vary widely. The values included in the standard GASWorkS Pipe Property Table are as recommended by the American Gas Association, *System Design* Handbook, *Book D-1*. The User may revise any of the standard values as desired. Pipe diameters must be entered as a Size/Type Code included in the Pipe Property Table for the roughness values in the Pipe Property Table to be used during the various computation routines.

Pipe Flow Equations

GASWorkS supports numerous industry-standard pipe flow equations, which calculate the flow and pressure drop of *Pipe* and *Fitting* Hydraulic Type features. Each equation is unique and may have individual limitations associated with its use. The User must determine the appropriate equation to use for the application being analyzed. Unless noted otherwise, the following base units and variable assignments apply to each equation description. Descriptions for the compressor, regulator, valve, well, and various supporting equations are described later in the *Appendix* section. The various equations supported by GASWorkS are described as follows.

Adjustment For Elevation

Where an elevation difference exists between the inlet and outlet of a pipe segment, and the normal pressure difference (ΔP) is a function of the difference between the squared inlet pressure and the squared outlet pressure ($\Delta P = P_1^2 - P_2^2$), the pressure difference (ΔP) term in the flow equation is replaced with the following...

$$\Delta P \text{ is replaced with, } \Delta P = P_1^2 - P_2^2 - SP_{AVE}^2$$

Where,
$$S = \frac{0.0375 SG(E_2 - E_1)}{T_f Z}$$

Reference 1

AGA-Fully Turbulent

The American Gas Association Fully Turbulent (AGA-Fully Turbulent) flow equation used by GASWorkS represents an equation with a variable roughness parameter. The equation is described as follows...

$$Q = 117.38 \times \frac{T_b}{P_b} \times \left(\frac{1}{f}\right)^{0.5} \times \left(\frac{\Delta P}{SG \times T_f \times L \times Z}\right)^{0.5} \times D^{2.5} \times E$$

Where,
$$\Delta P = P_1^2 - P_2^2$$
$$\left(\frac{1}{f}\right)^{0.5} = 4\log_{10}\left(\frac{3.7D}{\varepsilon}\right)$$

Reference 2

AGA-Partially Turbulent

The American Gas Association Partially Turbulent (AGA-Partially Turbulent) flow equation used by GASWorkS is described as follows...

$$Q = 117.38 \times \frac{T_b}{P_b} \times \left(\frac{1}{f} \right)^{0.5} \times \left(\frac{\Delta P}{SG \times T_f \times L \times Z} \right)^{0.5} \times D^{2.5} \times E$$

Where,

$$\Delta P = P_1^2 - P_2^2$$

$$\left(\frac{1}{f} \right)^{0.5} = 4\log_{10}\left(\frac{Re}{\left(\frac{1}{f} \right)^{0.5}} \right) - 0.6$$

Reference 2

AGA-Turbulent

The American Gas Association Turbulent (AGA-Turbulent) flow equation used by GASWorkS represents an equation with a fixed roughness parameter. The equation is described as follows...

$$Q = 1902.73 \times \frac{T_b}{P_b} \times \left(\frac{\Delta P}{SG \times T_f \times L \times Z} \right)^{0.5} \times D^{2.584} \times E$$

Where,

$$\Delta P = (P_1^2 - P_2^2)$$

Reference 3

Colebrook

The Colebrook-White (Colebrook) flow equation used by GASWorkS is described as follows...

$$Q = 117.38 \times \frac{T_b}{P_b} \times \left(\frac{1}{f}\right)^{0.5} \times \left(\frac{\Delta P}{SG \times T_f \times L \times Z}\right)^{0.5} \times D^{2.5} \times E$$

Where,

$$\Delta P = P_1^2 - P_2^2$$

And,

$$Re \leq 2000, \qquad \left(\frac{1}{f}\right)^{0.5} = 0.25(Re)^{0.5}$$

$$Re > 2000, \qquad \left(\frac{1}{f}\right)^{0.5} = -4\log_{10}\left(\frac{\varepsilon}{3.7D} + \frac{1.255}{Re}\left(\frac{1}{f}\right)^{0.5}\right)$$

Reference 2

IAPMO-High Pressure

The International Association of Plumbing and Mechanical Officials - High Pressure (IAPMO-HP) flow equation used by GASWorkS is described as follows...

$$Q = 741.2 \times \frac{T_b}{P_b} \times \frac{1}{\mu^{0.08223}} \times \left(\frac{\Delta P}{SG^{0.848} \times T_f \times L \times Z}\right)^{0.541} \times D^{2.625} \times E$$

Where,

$$\Delta P = P_1^2 - P_2^2$$

Reference 4

IAPMO-Low Pressure

The International Association of Plumbing and Mechanical Officials - Low Pressure (IAPMO-LP) flow equation used by GASWorkS is described as follows...

$$Q = 4619 \times \frac{T_b}{P_b} \times \frac{1}{\mu^{0.08223}} \times \left(\frac{\Delta P}{SG^{0.848} \times T_f \times L} \right)^{0.541} \times D^{2.625} \times E$$

Where,

$$\Delta P = P_1 - P_2$$

Reference 4

IFGC-High Pressure

The International Fuel Gas Code - High Pressure (IFGC-HP) flow equation used by GASWorkS is described as follows...

$$Q = 739.92 \times \frac{T_b}{P_b} \times \frac{1}{\mu^{0.08218}} \times \left(\frac{\Delta P}{SG^{0.848} \times T_f \times L \times Z} \right)^{0.5407} \times D^{2.6247} \times E$$

Where,

$$\Delta P = P_1^2 - P_2^2$$

Reference 9

IFGC-Low Pressure

The International Fuel Gas Code - Low Pressure (IFGC-LP) flow equation used by GASWorkS is described as follows...

$$Q = 4607.541 \times \frac{T_b}{P_b} \times \frac{1}{\mu^{0.08218}} \times \left(\frac{\Delta P}{SG^{0.848} \times T_f \times L} \right)^{0.5407} \times D^{2.6247} \times E$$

Where,

$$\Delta P = P_1 - P_2$$

Reference 9

IGE-TD3 General

The Institution of Gas Engineers Recommendation 3 - General (IGE-TD3 General) flow equation used by GASWorkS is described as follows...

$$Q = 117.3 \times \frac{T_b}{P_b} \times \left(\frac{1}{f} \right)^{0.5} \times \left(\frac{\Delta P}{SG \times T_f \times L \times Z} \right)^{0.5} \times D^{2.5} \times E$$

Where,

$$\Delta P = P_1^2 - P_2^2$$

$$\left(\frac{1}{f} \right)^{0.5} = 14.7519 + 3.5657X + 0.0362X^2$$

$$X = \log_{10}(Re) - 5$$

Reference 5

IGE3-Low Pressure

The Institution of Gas Engineers Recommendation 3 - Low Pressure (IGE3-LP) flow equation used by GASWorkS is described as follows...

$$Q = 165.94 \times \frac{T_b}{P_b^{0.5}} \times \left(\frac{1}{f}\right)^{0.5} \times \left(\frac{\Delta P}{SG \times T_f \times L \times Z}\right)^{0.5} \times D^{2.5} \times E$$

Where,

$$\Delta P = P_1 - P_2$$

$$\left(\frac{1}{f}\right)^{0.5} = 14.7519 + 3.5657X + 0.0362X^2$$

$$X = \log_{10}(Re) - 5$$

Reference 5

IGE3-Medium Pressure

The Institution of Gas Engineers Recommendation 3 - Medium Pressure (IGE3-MP) flow equation used by GASWorkS is described as follows...

$$Q = 117.3 \times \frac{T_b}{P_b} \times \left(\frac{1}{f}\right)^{0.5} \times \left(\frac{\Delta P}{SG \times T_f \times L \times Z}\right)^{0.5} \times D^{2.5} \times E$$

Where,

$$\Delta P = P_1^2 - P_2^2$$

$$\left(\frac{1}{f}\right)^{0.5} = 14.7519 + 3.5657X + 0.0362X^2$$

$$X = \log_{10}(Re) - 5$$

Reference 5

IGT-Improved

The Institute of Gas Technology - Improved (IGT-Improved) flow equation used by GASWorkS is described as follows...

$$Q = 664.3 \times \frac{T_b}{P_b} \times \frac{1}{\mu^{0.1111}} \times \left(\frac{\Delta P}{SG^{0.8} \times T_f \times L \times Z} \right)^{0.556} \times D^{2.667} \times E$$

Where,

$$\Delta P = P_1^2 - P_2^2$$

Reference 6

IMC-High Pressure

The International Mechanical Code - High Pressure (IMC-HP) flow equation used by GASWorkS is described as follows...

$$Q = 25987 \times \frac{1}{\mu^{0.08223}} \times \left(\frac{\Delta P}{SG^{0.848} \times T_f \times L} \right)^{0.541} \times D^{2.623} \times E$$

Where,

$$\Delta P = P_1^2 - P_2^2$$

Reference 7

IMC-Low Pressure

The International Mechanical Code - Low Pressure (IMC-LP) flow equation used by GASWorkS is described as follows...

$$Q = 162070 \times \frac{1}{\mu^{0.08223}} \times \left(\frac{\Delta P}{SG^{0.848} \times T_f \times L} \right)^{0.541} \times D^{2.623} \times E$$

Where,

$$\Delta P = P_1 - P_2$$

Reference 7

GASWorkS™ 10.1 *Appendix*

Mueller High Pressure

The Mueller High Pressure (Mueller-HP) flow equation used by GASWorkS is described as follows...

$$Q = 493.7 \times \frac{T_b}{P_b} \times \frac{1}{\mu^{0.15} \times SG^{0.425}} \times \left(\frac{\Delta P}{T_f \times L \times Z} \right)^{0.575} \times D^{2.725} \times E$$

Where,

$$\Delta P = P_1^2 - P_2^2$$

Reference 8

Mueller-Low Pressure

The Mueller Low Pressure (Mueller-LP) flow equation used by GASWorkS is described as follows...

$$Q = 735.4 \times \frac{T_b}{P_b} \times \frac{P_{ATM}^{0.575}}{\mu^{0.15} \times SG^{0.425}} \times \left(\frac{\Delta P}{T_f \times L} \right)^{0.575} \times D^{2.725} \times E$$

Where,

$$\Delta P = P_1 - P_2$$

Reference 8

Oliphant

The Oliphant flow equation used by GASWorkS is described as follows...

$$Q = 1492.8 \times \frac{T_b}{P_b} \times \left(\frac{\Delta P}{SG \times T_f \times L \times Z} \right)^{0.5} \times \left(D^{2.5} + \frac{D^3}{30} \right) \times E$$

Where,

$$\Delta P = P_1^2 - P_2^2$$

Reference 2

Panhandle-A

The Panhandle-A flow equation used by GASWorkS is described as follows...

$$Q = 1846.7 \times \left(\frac{T_b}{P_b}\right)^{1.0788} \times \left(\frac{\Delta P}{SG^{0.853} \times T_f \times L \times Z}\right)^{0.5392} \times D^{2.6182} \times E$$

Where,

$$\Delta P = P_1^2 - P_2^2$$

Reference 2

Panhandle-B

The Panhandle-B flow equation used by GASWorkS is described as follows...

$$Q = 2431 \times \left(\frac{T_b}{P_b}\right)^{1.02} \times \left(\frac{\Delta P}{SG^{0.961} \times T_f \times L \times Z}\right)^{0.51} \times D^{2.53} \times E$$

Where,

$$\Delta P = P_1^2 - P_2^2$$

Reference 2

Pole-Low Pressure

The Pole Low Pressure (Pole-LP) flow equation used by GASWorkS is described as follows...

$$Q = 617.7 \times \frac{T_b}{P_b} \times \left(\frac{1}{f}\right)^{0.5} \times \left(\frac{\Delta P}{SG \times T_f \times L}\right)^{0.5} \times D^{2.5} \times E$$

Where,

$$\Delta P = P_1 - P_2$$

$$\left(\frac{1}{f}\right)^{0.5} = 9.893589 + 2.183836 \ln(D)$$

Reference 3

Spitzglass-High Pressure

The Spitzglass High Pressure (Spitzglass-HP) flow equation used by GASWorkS is described as follows...

$$Q = 2209 \times \frac{T_b}{P_b} \times \left(\frac{1}{f}\right)^{0.5} \times \left(\frac{\Delta P}{SG \times T_f \times L \times Z}\right)^{0.5} \times D^{2.5} \times E$$

Where,

$$\Delta P = P_1^2 - P_2^2$$

$$\frac{1}{f} = \left(\frac{1}{1 + \dfrac{3.6}{D} + 0.03D}\right)$$

Reference 3

Spitzglass-Low Pressure

The Spitzglass Low Pressure (Spitzglass- LP) flow equation used by GASWorkS is described as follows...

$$Q = 11621 \times \frac{T_b}{P_b} \times \left(\frac{1}{f}\right)^{0.5} \times \left(\frac{\Delta P}{SG \times T_f \times L}\right)^{0.5} \times D^{2.5} \times E$$

Where,

$$\Delta P = P_1 - P_2$$

$$\frac{1}{f} = \left(\frac{1}{1 + \dfrac{3.6}{D} + 0.03D}\right)$$

Reference 3

TracPipe

This is a proprietary equation and constants provided by OmegaFlex Inc for use with their TracPipe branded pipe.

Weymouth

The Weymouth flow equation used by GASWorkS is described as follows...

$$Q = 1312.5 \times \frac{T_b}{P_b} \times \left(\frac{\Delta P}{SG \times T_f \times L \times Z} \right)^{0.5} \times D^{2.667} \times E$$

Where,

$$\Delta P = P_1^2 - P_2^2$$

Reference 2

Equation Variables

Unless otherwise noted, the following base units and variable definitions apply to all of the methods and equations described in the *Pipe Flow Equations* section:

D = Inside Pipe Diameter, inches
ε = Internal Pipe Wall Roughness, inches
E = Pipe Efficiency, percent / 100
E_1 = Pipe Inlet (Upstream) Elevation, feet
E_2 = Pipe Outlet (Downstream) Elevation, feet
f = Fanning Friction Factor, dimensionless
L = Pipe Length, feet
ΔP = Pressure Drop across a Pipe Segment, psia^2 or psia
P_1 = Pipe Inlet (Upstream) Pressure, psia
P_2 = Pipe Outlet (Downstream) Pressure, psia
P_{ATM} = Average Atmospheric Pressure, psia
P_{AVE} = Average Pressure along the Pipe Segment, psia
P_b = Base Pressure, psia
Q = Volumetric Flow Rate at the specified Base Pressure and Temperature, cfh
Re = Reynolds Number, dimensionless
S = Elevation Compensation Factor, dimensionless
SG = Specific Gravity, dimensionless
T_b = Base Temperature, Rankine
T_f = Average Gas Flowing Temperature, Rankine
μ = Absolute Viscosity, lbm/ft-sec
Z = Compressibility Factor, dimensionless

$P_1 = P_{1\_GAUGE} + P_{1\_ATM}$

P_{1_ATM} = Atmospheric Pressure at First Conditions, psia

P_{1_GAUGE} = Gauge Pressure at First Conditions, psig

$P_2 = P_{2\_GAUGE} + P_{2\_ATM}$

P_{2_ATM} = Atmospheric Pressure at Second Conditions, psia

P_{2_GAUGE} = Gauge Pressure at Second Conditions, psig

References

1) American Gas Association, *Steady Flow In Gas Pipelines*.

2) Gas Processors Suppliers Association, *Engineering Data Book*, Eleventh Edition - FPS, 1998.

3) Industrial Press, *Gas Engineers Handbook*, 1965.

4) International Association of Plumbing and Mechanical Officials, *Uniform Mechanical Code / Uniform Plumbing Code, 2006*.

5) (British) Institute Of Gas Engineers Standard - IGE/TD/3

6) Gas Age Magazine, *Gas Behavior In Distribution Systems,* May 1967.

7) International Code Council, *International Mechanical Code*, 1996.

8) American Gas Association, *Gas Engineering and Operating Practices - System Design*, GEOP Series Book D-1, 1990.

9) American Gas Association, *International Fuel Gas Code*, 2012.

Equation Application Reference

The following table provides application notes and comments for each equation supported by GASWorkS. This table represents only a portion of the full collection of equations for calculating pressure and flow in gas piping systems. Equations are added to GASWorkS based on User requests. If a User requires a specific equation and the equation may be documented, we may be willing to add the equation to the software.

Equation Name	Application Comments
AGA-Fully Turbulent	Applicable to transmission systems with fully turbulent flow conditions. Accounts for relative pipe roughness using the rough pipe law. This form of the equation allows input of the pipe wall roughness value. Depending on the pipe wall roughness it yields moderately conservative results compared to other transmission-style equations.
AGA-Partially Turbulent	Applicable to transmission systems with partially turbulent flow conditions. Accounts for relative pipe roughness using the smooth pipe law. Moderately conservative compared to other transmission-style equations.

Equation Name	*Application Comments*
AGA-Turbulent	Applicable to transmission systems with fully turbulent flow conditions. Uses a fixed relative pipe roughness with the rough pipe law. Moderately conservative compared to other transmission-style equations.
Colebrook	Applicable to most any piping applications. Often used for transmission systems. Applies the Moody friction factor to the Fundamental Gas Flow equation using the Colebrook-White approximation. Results vary based on the value of roughness.
IAPMO-HP	Applicable to fuel piping systems operating at 1.5 psig (10 kPa) and above. A high-pressure version of the Polyflo equation as adopted by the International Association of Plumbing and Mechanical Officials.
IAPMO-LP	Applicable to fuel piping systems operating at less than 1.5 psig (10 kPa). A low-pressure version of the Polyflo equation as adopted by the International Association of Plumbing and Mechanical Officials.
IFGC-HP	Applicable to fuel piping systems operating at 1.5 psig (10 kPa) and above. A high-pressure version of the Polyflo equation as implemented by the AGA & ICC International Fuel Gas Code.
IFGC-LP	Applicable to fuel piping systems operating at less than 1.5 psig (10 kPa). A low-pressure version of the Polyflo equation as implemented in the AGA & ICC International Fuel Gas Code.
IGE-TD3 General	Applicable to general distribution systems. Developed by the British Gas Engineers Institute. Recommended for pressures less than or equal to 101.5 psig (7 Bar).
IGE3-LP	Applicable to low-pressure distribution systems. Developed by the British Gas Engineers Institute. A modified version of the IGE3 - General equation for use at pressures less than or equal to 1.1 psig (75 mBar).
IGE3-MP	Applicable to medium-pressure distribution systems. Developed by the British Gas Engineers Institute. A modified version of the IGE3 - General equation for use at pressures greater than 1.1 psig (75 mBar) and less than or equal to 29 psig (2 Bar). Same as the IGE3-General equation.

Equation Name	*Application Comments*
IGT-Improved	Applicable to distribution systems, though often used for all types of systems operating between 1 to 500 psig. Yields moderate results. Developed in the 1960s. The Gas Engineers Handbook notes that this equation is widely used for 10 inch water column to 60 psig systems for 2-inch and larger diameter pipe. The AGA GEOP text reports that this equation applies to distribution systems where the Reynolds Numbers range between 16000 and 3000000 with 2% deviation from the smooth pipe law, and for 3 to 30-inch diameter pipe operating at inches water column, 1.5 to 20-inch diameter pipe operating between 2 and 20 psig, and 0.75 to 120-inch diameter pipe operating between 20 and 100 psig. Reynolds Number dependent.
IMC-HP	Applicable to fuel piping systems operating at 1.5 psig (10 kPa) and higher. A high-pressure version of the Polyflo equation as adopted by the International Code Council.
IMC-LP	Applicable to fuel piping systems operating at less than 1.5 psig (10 kPa). A low-pressure version of the Polyflo equation as adopted by the International Code Council.
Mueller-HP	Applicable to distribution systems. Yields non-conservative results. The Gas Engineers Handbook reports that the equation is accurate in "low" pressure systems for up to 8-inch pipe and in "medium" pressures for up to 2-inch pipe. The AGA GEOP text reports that this equation applies to distribution systems where the Reynolds Numbers range between 2000 and 125000 with 2% deviation from the smooth pipe law, and for 0.375 to 6-inch diameter pipe operating at inches water column, 0.375 to 2-inch diameter pipe operating between 2 and 20 psig, and 0.375 to 1.5-inch diameter pipe operating between 20 and 100 psig. Reynolds Number dependent.

Equation Name	*Application Comments*
Mueller-LP	Applicable to distribution systems. Yields non-conservative results. Derived from the Mueller - High Pressure equation as outlined in AGA GEOP text. Reynolds Number dependent.
Oliphant	Applicable to plant piping and gathering systems operating between vacuum and 100 psig. Yields conservative results. Not Reynolds Number dependent - diameter dependent.
Panhandle-A	Applicable to transmission systems. Yields moderate results. Developed in the 1940s. The Gas Engineers Handbook reports that the equation applies for large-diameter transmission piping where the Reynolds Numbers range between 5 and 20 million. The Handbook also suggests an average efficiency for steel pipelines of 0.92 for this equation. The GPSA Data Book reports that this equation is intended to reflect the flow of gas through smooth pipes. When used with an efficiency of 0.90, the equation reasonably approximates the partially turbulent general flow equation. The equation becomes less accurate as the flow rate increases. An efficiency of 0.92 is often used with this equation. The AGA GEOP text reports that this equation applies to distribution systems where the Reynolds Numbers range between 1300000 and 75000000 with a 2% deviation from the smooth pipe law, and a 16-inch and larger diameter pipe operating for 20 psig and higher. Reynolds Number dependent.

Equation Name	*Application Comments*
Panhandle-B	Applicable to transmission systems. A revised version of Panhandle-A was published in 1956. Depending on the flow rate, it may be the least conservative transmission equation. The Gas Engineers Handbook notes that the equation applies for large-diameter transmission piping where the Reynolds Numbers range between 5 and 20 million. The Handbook suggests an average efficiency for steel pipelines of 0.90 for this equation. The GPSA Data Book notes that this equation more nearly approximates the fully turbulent general flow equation. The equation will underestimate flow rates at low flow rates and overestimate flow rates at high flow rates. Efficiency factors of 0.88 to 0.94 are often used with this equation. Reynolds Number dependent.
Pole-LP	Applicable to fuel and yard line piping operating below 3 psig. Yields conservative results. The Gas Engineers Handbook notes that this equation is widely used for 4 to 10 inch water column pressure systems for 2 to 4-inch diameter pipe. The equation is applicable with effective pipe roughness values of up to ten times that of clean steel pipe. Not Reynolds Number dependent; constant friction/transmission factor.
Spitzglass-HP	Applicable to distribution systems operating below 100 psig. Yields conservative results. The Gas Engineers Handbook reports that this equation is widely used for up to 10-inch diameter pipe and is applicable with effective pipe roughness values of up to ten times that of clean steel pipe. Not Reynolds Number dependent - diameter dependent.

Equation Name	*Application Comments*
Spitzglass-LP	Applicable to low-pressure distribution systems operating below 3 psig. Yields moderate results. The Gas Engineers Handbook reports that this equation is widely used for systems operating under 1 psig. The AGA GEOP text reports that this equation was developed from flow tests on large-diameter cast iron pipes operated at inches water column pressures and partially turbulent flow conditions. The equation was originally used in flow calculations in low-pressure cast iron systems. The text also reports that the equation yields extremely conservative results for 16-inch and larger diameter pipes and yields un-conservative results for pipes sizes less than 16-inch. Not Reynolds Number dependent - diameter dependent.
TracPipe	Applies only to Corrugated Stainless Steel Tubing manufactured by OmegaFlex Inc. using the brand name TracPipe.
Weymouth	Applicable to transmission systems, though often used for distribution, gathering, and transmission systems. The equation yields conservative results. It was published in 1912. The Gas Engineers Handbook reports that this equation provides a reasonable approximation of the general rough pipe equation for a diameter equal to 10 inches and effective pipe wall roughness equal to 0.002 inches. The Handbook also suggests that the equation is widely used for 3 to 20 psig systems. Also, the Handbook suggests an average efficiency for steel pipelines of 1.10 for this equation. The GPSA Data Book reports that for short pipelines and gathering systems, this equation agrees closely with metered volumes. However, the degree of error increases with pressure. The AGA GEOP text reports that the applicable efficiency factor for this equation is 1.00 for a 36-inch diameter pipe and can range from 1.10 to 2.00 for a 30-inch and smaller pipe in transmission service. The text suggests that the efficiency factor ranges between 1.1 and 1.2 for 2 to 6-inch diameter pipe in distribution service. The text further suggests that the equation does not apply to calculations in distribution systems. Not a Reynolds Number dependent equation.

Equation Name	Application Comments
References	

References
• American Gas Association, *Gas Engineering and Operating Practices - System Design*, GEOP Series Book D-1, 1990.
• American Gas Association, *Steady Flow In Gas Pipelines*, Technical Report No. 10.
• British Gas Engineers Institute, Standard IGE/TD/3, Recommendations On Transmission And Distribution Practice, Edition 4, 2003.
• Crane Technical Paper No. 410, *Flow Of Fluids Through Valves, Fittings, And Pipe*.
• Gas Processors Suppliers Association, *Engineering Data Book*, Volume II.
• Gulf Publishing Company, *Pipeline Rules of Thumb Handbook*.
• Industrial Press, *Gas Engineers Handbook*, 1965.
• International Association of Plumbing and Mechanical Officials, *Uniform Mechanical Code*, 2006.
• International Code Council, *International Mechanical Code*, 1996.

Recommendations

As seen by the number and variety of flow equations listed in the previous table, there seems to be no certain flow equation that accurately reflects all of the conditions that may be present in all types of gas piping systems. There is not one equation that meets all requirements, or there wouldn't be such a variety to choose from. For that reason, the User must ultimately compare their specific application to each equation's performance and select an appropriate equation. Further, it is extremely difficult to recommend a flow equation because of the variety of applications that Users might encounter. However, generally, we make the following recommendations - ***use at your own risk***:

Application	Equation
Low Pressure Distribution (less than 3 psig)	Spitzglass-LP
Medium and High Pressure Distribution (3 to 100 psig)	IGT-Improved
Low Pressure Transmission (100 to 300 psig)	IGT-Improved
High Pressure Transmission (300 psig and greater)	Panhandle-A
Vacuum Gathering	Oliphant
Low Pressure Gathering (less than 100 psig)	IGT-Improved
High Pressure Gathering (100 psig and greater)	IGT-Improved
Vacuum Plant Piping	Oliphant

Application	Equation
Low Pressure Plant or Fuel Piping (less than 1.5 psig)	IAPMO-LP IMC-LP
Medium Pressure Plant or Fuel Piping (1.5 to 10 psig)	IAPMO-HP IGT-Improved IMC-HP
High Pressure Plant or Fuel Piping (10 to 100 psig)	IGT-Improved Oliphant

Regulator Flow Equations

GASWorkS supports calculating the various values associated with the flow through a regulator using a variety of industry and manufacturer equations. The various equations supported by GASWorkS are described in the following sections.

American

The American Regulator flow equation used by GASWorkS is described as follows...

$$If \; \frac{P_1}{P_2} < 1.894, \qquad Q = F_B \times C \times \sqrt{\frac{P_2(P_1 - P_2)}{SG}}$$

$$If \; \frac{P_1}{P_2} \geq 1.894, \qquad Q = \frac{F_B \times C \times P_1}{2} \times \sqrt{\frac{1}{SG}}$$

$$F_B = \left(\frac{T_B}{520}\right) \times \left(\frac{14.7}{P_B}\right)$$

Reference 1

American AFV

The American AFV Regulator flow equation used by GASWorkS is described as follows. This equation is suitable for use with American Axial Flow Valve regulators. *Note* - The American AFV equation is essentially the same as the ISA -S75.01 equation.

$$If\ (P_1 - P_2) < (F_k X_t P_1), \quad Q = 1359.792 \times F_B \times C_V \times Y \times \sqrt{\frac{P_1^2 - P_1 \times P_2}{SG \times T_F}}$$

$$Y = 1 - \frac{P_1 - P_2}{3 \times F_k \times X_t \times P_1}$$

$$If\ (P_1 - P_2) \geq (F_k X_t P_1), \quad Q = 906.981 \times F_B \times C_V \times P_1 \times \sqrt{\frac{F_k \times X_t}{SG \times T_F}}$$

$$F_B = \left(\frac{T_B}{520}\right) \times \left(\frac{14.73}{P_B}\right)$$

Reference 2

Equimeter

The Equimeter Regulator flow equation used by GASWorkS is described as follows. This equation is suitable for use with certain models of Equimeter, Sensus, and Rockwell regulators.

$$If\ \frac{P_1}{P_2} < 1.894, \quad Q = F_B \times K \times \sqrt{\frac{0.6 P_2 (P_1 - P_2)}{SG}}$$

$$If\ \frac{P_1}{P_2} \geq 1.894, \quad Q = \frac{F_B \times K \times P_1}{2} \times \sqrt{\frac{0.6}{SG}}$$

$$F_B = \left(\frac{T_B}{520}\right) \times \left(\frac{14.65}{P_B}\right)$$

Reference 3, 4

Fisher

The Fisher Regulator flow equation used by GASWorkS is described as follows. This equation is suitable for use with certain Fisher Controls regulator models.

$$If \ \frac{P_2}{P_1} > 0.5, \qquad Q = F_B \times C_g \times P_1 \times \sqrt{\frac{520}{SG \times T_F}} \times \sin\left(\frac{59.638}{C_1} \sqrt{\frac{P_1 - P_2}{P_1}} \right)$$

$$If \ \frac{P_2}{P_1} \leq 0.5, \qquad Q = F_B \times C_g \times P_1 \times \sqrt{\frac{520}{SG \times T_F}}$$

$$F_B = \left(\frac{T_B}{520} \right) \times \left(\frac{14.7}{P_B} \right)$$

Reference 5

Grove 80

The Grove 80 Regulator flow equation used by GASWorkS is described as follows. This equation is suitable for use with Grove Model 80 regulators.

$$\text{If } \frac{P_1 - P_2}{P_1} < (F_k \, X_t), \qquad Q = 1359 \times Y_1 \times \sqrt{\frac{P_1(P_1 - P_2)}{SG \times T_f \times Z}}$$

$$\text{If } \frac{P_1 - P_2}{P_1} \geq (F_k \, X_t), \qquad Q = 22.8 \times C_1 \times C \times P_1 \times \sqrt{\frac{F_k}{SG \times T_f \times Z}}$$

Where,

$C = $ *Partially Open Capacity Factor...*

$$C = 0, \qquad\qquad\qquad P_1 - P_2 \leq 0.8E$$

$$C = C_p \left(\frac{(P_1 - P_2) - 0.8E}{1.2E} \right), \qquad 0.8E < P_1 - P_2 < 2E$$

$$C = C_p, \qquad\qquad\qquad P_1 - P_2 \geq 2E$$

$C_1 = $ *Critical Flow Factor...*

$$C_1 = 9.253 + 49.171X_t - 27.268X_t^2 + 8.586X_t^3$$

$C_p = $ *Valve Wide Open Capacity Coefficient*

$E = $ *Tube Expansion Factor*

$F_k = $ *Specific Heat Ratio Factor...*

$$F_k = \frac{k}{1.4}$$

$Y_1 = $ *Expansion Coefficient...*

$$Y_1 = 1, \qquad\qquad\qquad \frac{P_1 - P_2}{P_1} \leq 0.02$$

$$Y_1 = 1 - \left(\frac{0.333(P_1 - P_2)}{X_t \times P_1 \times F_k} \right), \qquad 0.02 < \frac{P_1 - P_2}{P_1} < (X_t \times F_k)$$

$$Y_1 = 0.667, \qquad\qquad\qquad \frac{P_1 - P_2}{P_1} \geq (X_t \times F_k)$$

Continued on following page...

Continued previous page...

X_t = *Critical Pressure Drop Ratio*

Z = *Compressibility Factor...*
  If the Specific Gravity is between 0.5 *and* 0.6,
  interpolated from manufacturer's literature,
  otherwise equal 1.0

Reference 2

Grove 900

The Grove 900 Regulator flow equation used by GASWorkS is described as follows. This equation is suitable for use with Grove model 900TE Flexflo regulators.

$$Q = 1359 F_B \times C_V \times A \times Y \times \sqrt{\frac{DP}{SG \times T_F \times Z}}$$

$$\text{If } \frac{P_1 - P_2}{P_1} < X_t F_k, \qquad DP = P_1(P_1 - P_2)$$

$$Y = 1 - \frac{P_1 - P_2}{3 X_t F_k P_1}$$

$$\text{If } \frac{P_1 - P_2}{P_1} \geq X_t F_k, \qquad DP = X_t F_k$$

$$Y = 0.667$$

$$F_B = \left(\frac{T_B}{520}\right) \times \left(\frac{14.7}{P_B}\right)$$

Where,

C_V = *Valve Capacity Factor*

X_t = *Critical Pressure Drop Ratio*

Z = *Supercompressibility Factor...*
If the specific gravity is between 0.5 *and* 0.6, *interpolated from manufacturer's literature, otherwise equal* 1.0.

Reference 6, 7, 8

ISA-S75.01

This method uses the equations described in the Instrument Society of America Standard ISA-S75.01-1985, "Flow Equations for Sizing Control Valves," Revised 1995. The User may refer to Reference 7 for further information on calculating the regulator parameters using this method. *Note* - The ISA -S75.01 equation is essentially the same as the American AFV equation.

This method is suitable for use with regulators that are stated as being compliant with this standard and providing suitable sizing parameters.

Itron

The Itron Regulator flow equation used by GASWorkS is described as follows. This equation is suitable for use with certain models of Itron regulators.

$$If \; \frac{P_1}{P_2} < 1.894, \qquad Q = F_B \times K \times \sqrt{\frac{0.6P_2(P_1 - P_2)}{SG}}$$

$$If \; \frac{P_1}{P_2} \geq 1.894, \qquad Q = \frac{F_B \times K \times P_1}{2} \times \sqrt{\frac{0.6}{SG}}$$

$$F_B = \left(\frac{T_B}{520} \right) \times \left(\frac{14.7}{P_B} \right)$$

Reference 9

Mokveld

The Mokveld Regulator flow equation used by GASWorkS is described as follows. This equation is suitable for use with Mokveld Valve regulators.

$$Q = 832.9438 \times F_B \times F_L \times C_V \times P_1 \times \left(Y - 0.148 Y^3\right) \times \frac{1}{\sqrt{SG \times T_F}}$$

$$Y = \frac{1.63}{F_L} \times \sqrt{\frac{P_1 - P_2}{P_1}}$$

$$Y \leq 1.5$$

$$F_B = \left(\frac{T_B}{518.67}\right) \times \left(\frac{14.69}{P_B}\right)$$

Where,

$$C_V = Valve \; Capacity \; Coefficient, \; Cfh/Psi$$

Reference 10

Mooney

The Mooney Regulator flow equation used by GASWorkS is described as follows. This equation is suitable for use with Mooney Controls regulators.

$$If \; \frac{P_1 - P_2}{P_2} < 0.64, \qquad Q = F_B \times C_g \times P_1 \times \sqrt{\frac{520}{SG \times T_F}} \times \sin\left(\frac{59.638}{C_1} \sqrt{\frac{P_1 - P_2}{P_1}}\right)$$

$$If \; \frac{P_1 - P_2}{P_2} \geq 0.64, \qquad Q = F_B \times C_g \times P_1 \times \sqrt{\frac{520}{SG \times T_F}}$$

$$F_B = \left(\frac{T_B}{520}\right) \times \left(\frac{14.7}{P_B}\right)$$

Reference 11

Pietro Fiorentini

The Pietro Fiorentini Regulator flow equation used by GASWorkS is described as follows. This equation is suitable for use with certain model Pietro Fiorentini regulators.

$$If \ \frac{P_1}{P_2} < 2, \qquad Q = 22.77972 \times F_B \times C_g \times P_1 \times \sqrt{\frac{1}{SG \times T_F}} \times \sin\left(K_1 \sqrt{\frac{P_1 - P_2}{P_1}} \right)$$

$$If \ \frac{P_1}{P_2} \geq 2, \qquad Q = 22.77972 \times F_B \times C_g \times P_1 \times \sqrt{\frac{1}{SG \times T_F}}$$

$$F_B = \left(\frac{T_B}{519} \right) \times \left(\frac{14.692}{P_B} \right)$$

Reference 12

Rockwell

The Rockwell flow equation used by GASWorkS is described as follows...

$$If \ \frac{P_1}{P_2} < 1.894, \qquad Q = F_v \times \sqrt{\frac{0.6 P_2 (P_1 - P_2)}{SG}}$$

$$If \ \frac{P_1}{P_2} \geq 1.894, \qquad Q = \frac{F_v \times P_1}{2} \times \sqrt{\frac{0.6}{SG}}$$

$$Where,$$
$$F_v = Regulator \ Valve \ Constant$$

Reference 3, 4

Table Based Calculation

GASWorkS allows the capacity of a regulator to be defined by a special format, the Regulator Sizing Table. The table can be created from capacity tables, charts, or graphs published by the manufacturer. Devices that use table-based capacity values are restricted to capacity calculations only. The inlet and outlet pressure values must fall within the values reported in the table. When the pressure values are within the included data, the capacity is interpolated from the available values. A calculation error will occur when the pressure values fall outside the limits of the data contained in the table - extrapolation is not supported. The interpolation method used by the table-based calculation method is described in the GASCalc Value Interpolation Calculation Reference. The required tables can be created using the Create/Edit Regulator Sizing Table routine in the GASCalc software.

Universal

The Universal flow equation used by GASWorkS is described as follows...

$$If \ \frac{P_1 - P_2}{P_2} < 0.64, \qquad Q = C_g \times P_1 \times \sqrt{\frac{520}{SG \times T_f}} \times \sin\left(\frac{59.638}{C_1}\sqrt{\frac{P_1 - P_2}{P_1}}\right)$$

$$If \ \frac{P_1 - P_2}{P_2} \geq 0.64, \qquad Q = C_g \times P_1 \times \sqrt{\frac{520}{SG \times T_f}}$$

Where,
C_g = *Gas Sizing Coefficient*
C_1 = *Valve Recovery Coefficient* – *assumed constant for a specific model*

Reference 5

Equation Variables

Unless otherwise noted, the following base units and variable definitions apply to all of the methods and equations described in the *Regulator Flow Equations* section:

k = Specific Heat Ratio, dimensionless
P_1 = Pipe Inlet (Upstream) Pressure, psia
P_2 = Pipe Outlet (Downstream) Pressure, psia
Q = Volumetric Flow Rate at the specified Base Pressure and Temperature, cfh
SG = Specific Gravity, dimensionless
T_f = Average Gas Flowing Temperature, Rankine

$P_1 = P_{1\_GAUGE} + P_{1\_ATM}$

P_{1_ATM} = Atmospheric Pressure at First Conditions, psia

P_{1_GAUGE} = Gauge Pressure at First Conditions, psig

$P_2 = P_{2\_GAUGE} + P_{2\_ATM}$

P_{2_ATM} = Atmospheric Pressure at Second Conditions, psia

P_{2_GAUGE} = Gauge Pressure at Second Conditions, psig

References

1) American Meter Company, Regulator Bulletin - Model 1200, SB-8505.1.

2) American Meter Company, *Axial Flow Valves Capacity Tables,* TDB 9610.5.

3) Equimeter Inc, Bulletin Model 441-57S, R-1360 Rev 4.

4) Rockwell International, Bulletin Model 441-57S, R 1360 Rev 3.

5) Fisher Controls, *Catalog 10 - Sizing and Selection Data*.

6) Grove Model 900 TE, Bulletin 900 TE 1 R1, 8/94.

7) Instrument Society of America Standard ISA-S75.01-1985, "Flow Equations for Sizing Control Valves," Revised 1995.

8) Grove Flexsize Regulator Sizing Software, Version 3.0.0.1, 2003.

9) Itron Gas, CL38 Regulator, Publication 101077SP-01, 2010.

10) Mokveld Valves, *Self Acting Regulators*, 2012.

11) Mooney Controls, *Sizing - Compressible Gases*.

12) Pietro Fiorentini, *Sizing Pressure Regulators & Control Valves*, 2010.

Standard Dimensional Units

The following table describes the standard or "native" units used by GASWorkS in its Solution and Calculation routines. All other units are converted to these native units during the computations and then back to the original units when the computation is complete.

Parameter Type	Standard Dimensional Units
Coordinates	Feet
Customer Load	Cfh
Energy	Therms
Heating Value	Btu/cf
Node Pressure	Psi
Node Load	Cfh
Node Elevation	Feet
Node Temperature	Rankine
Other Temperatures	Rankine
Other Pressures	Psi
Pipe Diameter	Inches
Pipe Efficiency	Decimal
Pipe Flow Rate	Cfh
Pipe Length	Feet
Pipe Roughness	Inches
Pressure Drop	Psi
Velocity	Feet/sec
Viscosity	Lbm/Ft-sec

Standard Property Tables

The "standard" Property Tables included in the GASWorkS software provide a basis for building your Property Tables. The values contained in the Property Tables may be modified, or new items may be added. The individual tables may be combined to create a custom Property Table. The general contents of the various standard Property Tables are described in the following tables. The specific description and source reference for each Property Table item is included in the associated *Property Tables Editor* topic.

Note that new items and Property Tables are occasionally added as Users request new or additional items. The Property Table Report and content included with your version of the software may vary from the Property Tables described on the following pages. New and revised Property Tables may be downloaded from our website, www.b3pellc.com.

Compressor Property Table	
Table Name	**Content**
compressor.cpf	Contains definitions for generic *Compressor* Hydraulic Types.

Fitting Property Table	
Table Name	**Content**
fitting.fpf	Contains a comprehensive list of definitions for fittings and valves ranging from ½" to 60" in size.
fitting_dp.fpf	Contains example definitions of fixed pressure drop style fittings.
fitting_efv_dresser.fpf	Contains definitions for excess flow valves provided by Dresser Inc.
fitting_efv_prefection.fpf	Contains definitions for excess flow valves provided by Perfection Corporation (Elster Perfection).
fitting_efv_umac.fpf	Contains definitions for excess flow valves provided by UMAC (GasBreaker).
fitting_pe_punch_tee.fpf	Contains a list of definitions for polyethylene punch tees.
fitting_us_large.fpf	Contains a list of definitions for fittings and valves ranging from 14" to 60" in size.
fitting_us_small.fpf	Contains a list of definitions for fittings and valves ranging from ½" to 12" in size.
uk_fitting.fpf	Contains a list of definitions for plastic and steel fittings commonly used in the UK.

Fitting Property Table	
Table Name	**Content**
ukdm_fitting.fpf	Contains a list of definitions for special fittings used by the UK Design Module.

Pipe Property Table	
Table Name	**Content**
pipe.ppf	Contains a comprehensive list of definitions for plastic and steel pipes from ½" to 48" in size.
pipe_cts_acr_nfgc.ppf	Contains a list of definitions for copper tubing based on the ACR sizing system as listed in the AGA/NFPA National Fuel Gas Code.
pipe_cts_k&l_nfgc.ppf	Contains a list of definitions for copper tubing based on the K&L sizing system as listed in the AGA/NFPA National Fuel Gas Code.
pipe_cts_k_nfgc.ppf	Contains a list of definitions for copper tubing based on the K sizing system as listed in the AGA/NFPA National Fuel Gas Code.
pipe_cts_l_nfgc.ppf	Contains a list of definitions for copper tubing based on the L sizing system as listed in the AGA/NFPA National Fuel Gas Code.
pipe_dns.ppf	Contains a list of definitions for steel and polyethylene pipe sizes using the Nominal Diameter sizing specifications.
pipe_hdpe_ips.ppf	Contains a list of definitions for high-density polyethylene pipe sizes using the IPS convention.
pipe_mdpe_cts.ppf	Contains a list of definitions for medium-density polyethylene pipe sizes using the CTS convention.
pipe_mdpe_ips.ppf	Contains a list of definitions for medium-density polyethylene pipe sizes using the IPS convention.
pipe_pe_cts.ppf	Contains a list of definitions for polyethylene pipe sizes using the IPS convention.
pipe_pe_dns.ppf	Contains a list of definitions for polyethylene pipes using the Nominal Diameter sizing specifications.
pipe_pe_nfgc.ppf	Contains a list of definitions for polyethylene pipe based on sizes listed in the AGA/NFPA National Fuel Gas Code.

Pipe Property Table	
Table Name	**Content**
pipe_pe_uk.ppf	Contains a list of definitions for polyethylene pipe based on common UK sizing specifications.
pipe_pe_us.ppf	Contains a list of definitions for polyethylene pipe based on common US sizing specifications.
pipe_steel_dns.ppf	Contains a list of definitions for steel pipe sizes using the Nominal Diameter sizing specifications.
pipe_steel_ips_large.ppf	Contains a list of definitions for steel pipe sizes using the IPS convention for sizes 14" to 42".
pipe_steel_ips_small.ppf	Contains a list of definitions for steel pipe sizes using the IPS convention for sizes ½" to 12".
pipe_steel_nfgc.ppf	Contains a list of definitions for steel pipe as listed in the AGA/NFPA National Fuel Gas Code.
pipe_uk.ppf	Contains a list of definitions for plastic, steel, cast, ductile, and spun iron pipe commonly used in the UK.
pipe_us.ppf	Contains a list of definitions for plastic, steel, cast, ductile, and spun iron pipe commonly used in the US.

Regulator Property Table	
Table Name	**Contents**
regulator.rpf	Contains a list of definitions for generic models and specific data for a selected group of models from various manufacturers.
regulator_fisher.rpf	Contains a list of definitions for a selected group of Fisher regulators.
regulator_generic.rpf	Contains a list of definitions for generic models based on various manufacturer's flow equations.
regulator_grove.rpf	Contains a list of definitions for a selected group of Grove regulators.
regulator_mooney.rpf	Contains a list of definitions for a selected group of Mooney regulators.
regulator_rockwell.rpf	Contains a list of definitions for a selected group of Rockwell regulators.

Valve Property Table	
Table Name	**Contents**
valve.vpf	Contains a list of definitions for generic models and specific data for a selected group of models from various manufacturers.

Well Property Table	
Table Name	**Contents**
well.wpf	Contains a definition for a single example/sample well.

Support Equations

GASWorkS uses several equations to support its various flow, pressure drop, and power equations. These support equations are described in the following sections.

Average Pressure

Some computations made by GASWorkS require determining the average pressure value along a pipe segment. When required, the average pressure is calculated as follows...

$$P_{AVE} = \frac{2}{3} \left(P_1 + P_2 - \frac{P_1 P_2}{P_1 + P_2} \right)$$

Equation Variables

The following base units and variable definitions apply to this equation:

P_1 = Pipe Inlet (Upstream) Pressure or First Specified Pressure, psia
P_2 = Pipe Outlet (Downstream) Pressure or Second Specified Pressure, psia
P_{AVE} = Average Pressure, psia

$P_1 = P_{1\_GAUGE} + P_{1\_ATM}$
P_{1_ATM} = Atmospheric Pressure at First Conditions, psia
P_{1_GAUGE} = Gauge Pressure at First Conditions, psig

$P_2 = P_{2\_GAUGE} + P_{2\_ATM}$
P_{2_ATM} = Atmospheric Pressure at Second Conditions, psia
P_{2_GAUGE} = Gauge Pressure at Second Conditions, psig

References

1) Gas Processors Suppliers Association, *Engineering Data Book*, 1998.

Energy To Volume Conversion

When load values are entered in energy type units (Kilowatt, Btu/hr, etc) they are converted to volumetric flow rates as follows...

$$Q = \frac{X \times Q_E}{HV}$$

Equation Variables

The following base units and variable definitions apply to this equation:

HV = Volumetric Heating Value of the Gas, btu/cf
Q = Volumetric Flow Rate
Q_E = Energy Usage Rate, Dimensional Units Varies
X = Unit Conversion Factor,
       Kilowatt = 3415.2
       Btu/hr = 1
       Mbtu/hr = 1,000

References

1) None - Derived.

Pipe Flow Velocity

The pipe flow velocities reported by GASWorkS are calculated as follows...

$$V_{Feet/\text{sec}} = 0.05093Q \times \frac{P_b \times T_{AVE}}{P \times T_b \times D^2}$$

Equation Variables

The following base units and variable definitions apply to this equation:

D = Inside Diameter of Tap Opening, inches
P = Lowest Pressure along the Pipe for Maximum Velocity values
P_b = Base Pressure, psia
Q = Volumetric Flow Rate at the specified Base Pressure and Temperature, cfh
T_{AVE} = Average Gas Flowing Temperature, Rankine
T_b = Base Temperature, Rankine
$V_{feet/sec}$ = Pipe Flow Velocity, feet/sec

References

1) None - Derived.

Pipe Hoop Stress

The pipe Hoop Stress reported by GASWorkS is calculated as follows...

$$S_{HOOP} = \frac{P \times D}{2t}$$

Equation Variables

The following base units and variable definitions apply to this equation:

D = Diameter of the Pipe Segment as defined below, inches
P = Operating Pressure of the Pipe Segment, psig
S_{HOOP} = Hoop Stress on the Pipe Segment, psi
t = Wall Thickness for the Pipe Segment, inches

Reference 1 - D = Outside Diameter for Traditional Industry Method
Reference 2 - D = Inside Diameter for Theoretically Correct Method
Reference 3 - D = Nominal Diameter for Occasional Industry Method
Reference 4 - D = Average Diameter for API 1102 Method

References

1) American Gas Association, *Gas Engineering and Operating Practices - System Design*, GEOP Series Book D-1, 1990.
2) Industrial Press, *Gas Engineers Handbook*, 1965.
3) None - Derived.
4) American Petroleum Institute, *Steel Pipelines Crossing Railroads and Highways*, API Recommended Practice 1102, Sixth Edition, 1993.

GASWorkS™ 10.1

Pipe Volume

The pipe volume reported by GASWorkS is calculated as follows...

$$V = \frac{\pi \, D^2}{576} \times \frac{P_{AVE}}{P_b} \times \frac{T_b}{T_{AVE}} \times \frac{1}{Z_{AVE}} \times L$$

Equation Variables

The following base units and variable definitions apply to this equation:

D = Inside Diameter or Equivalent Diameter of the Pipe Segment(s), inches
L = Length or Equivalent Length of the Pipe Segment(s), feet
P_{AVE} = Average Pressure, psia
P_b = Base Pressure, psia
π = 3.141592, Pi
T_b = Base Temperature, Rankine
T_{AVE} = Average Temperature, Rankine
V = Amount of Gas contained in the Segment(s) at the Average Pressure and Temperature, adjusted
      to the Base Pressure and Temperature, cf
Z_{AVE} = Average Compressibility, dimensionless

$P_{AVE} = P_{AVE\_GAUGE} + P_{ATM}$
P_{ATM} = Average Atmospheric Pressure, psia
P_{AVE_GAUGE} = Average Gauge Pressure, psig

References

1) Gas Processors Suppliers Association, *Engineering Data Book*, 1998.

Reynolds Number

The Reynolds Number, as used by GASWorkS, is described as follows...

$$Re = 0.01146 \times \frac{P_b}{T_b} \times \frac{Q \times SG}{D \times \mu}$$

Equation Variables

The following base units and variable definitions apply to this equation:

D = Inside Diameter of Tap Opening, inches
P_b = Base Pressure, psia
Q = Volumetric Flow Rate at the specified Base Pressure and Temperature, cfh
Re = Reynolds Number, dimensionless
SG = Specific Gravity, dimensionless
T_b = Base Temperature, Rankine
μ = Absolute Viscosity, lbm/ft-sec

References

1) American Gas Association, *Gas Engineering and Operating Practices - System Design*, GEOP Series Book D-1, 1990.

Specific Heat Values

Some computations made by GASWorkS require using either the Specific Heat Ratio or an individual Specific Heat Value. When required, these values are calculated as follows...

$$k = \frac{C_P}{C_V}$$

When C_P is required,

$$C_P = \frac{1.99k}{28.9625\,SG(k - 1)}$$

Equation Variables

The following base units and variable definitions apply to this equation:

C_P = Specific Heat at Constant Pressure, btu/lbm-degree Fahrenheit
C_V = Specific Heat at Constant Volume, btu/lbm-degree Fahrenheit
k = Specific Heat Ratio, dimensionless
SG = Specific Gravity (Ideal Gas Basis), dimensionless

References

1) McGraw-Hill, *De Laval Engineering Handbook*.

GASWorkS™ 10.1

Appendix

Table Of Contents

Supported Dimensional Units

GASWorkS uses a "standard" set of dimensional units for its internal calculations. When a value expressed in a different unit is encountered, it is converted internally for GASWorkS. The following table lists the supported dimensional units and the conversion factors used for internal use.

Unit Group	*Supported Types*	*Conversion Factor*
Coordinates Base Units = **Feet**	**Centimetre**	0.03281
	Degrees L/L (Degrees Longitude/Latitude)	N/A - Requires the use of the Projection functions.
	Degrees Web L/L (Degrees Longitude/Latitude using the "Web" Mercator coordinate system)	N/A - Requires the use of the Projection functions.
	Feet	1.0
	Inches	1/12
	Metres	3.281
Diameter, Roughness (Pipe) Base Units = **Inches**	**Centimetre, cm**	0.3937
	Feet, **ft**	12
	Inches, **in**	1
	Metres, **m**	39.37
	Microns	0.00003937
	Millimetre, mm	0.03937
Efficiency (Pipe) Base Units = **Decimal**	**Decimal**	1
	Percent, %	0.01
Elevation (Pipe) Base Units = **Feet**	**Feet**, **ft**	1
	Metres, **m**	3.281
Energy Base Units = **Therms**	**Btu** (British thermal unit)	0.00001
	kWh (thousands of watt hours)	0.03143
	MBtu (thousands of British Thermal Units)	0.01

B^3 B3PE LLC

Engineering & Software Services

Revision - 008, Copyright 2024, All Rights Reserved.

646

Unit Group	*Supported Types*	*Conversion Factor*
	MMBtu (millions of British Thermal Units)	10
	Therms	1
Flow Rate (Pipe) Load (Customer, Node) Base Units = **Cfh**	**Btu/hr** (British thermal units per hour)	See the *Support Equations* topic in this section for more information.
	Ccf/mo (hundred cubic feet per month)	0.13689
	Cfd (cubic feet per day)	0.041667
	Cfh (cubic feet per hour)	1
	Cfm (cubic feet per minute)	60
	Kilowatts, **kW**	See the *Support Equations* topic in this section for more information.
	Litres/min (litres per minute)	2.1186
	M3/d, **M3d** (cubic metres per day)	1.47125
	M3/hr, **M3h** (cubic metres per hour)	35.31
	MBtu/hr (thousands of British thermal units per hour)	See the *Support Equations* topic in this section for more information.
	Mcf/mo (thousands of cubic feet per month)	1.3689
	Mcfd (thousands of cubic feet per day)	41.667
	Mcfh (thousands of cubic feet per hour)	1000
	Mm3/d, **Mm3d** (thousands of cubic metres per day)	1471.25
	Mm3h (thousands of cubic metres per hour)	35310

Unit Group	Supported Types	Conversion Factor
	MMBtu/hr (millions of British thermal units per hour)	See the *Support Equations* topic in this section for more information.
	MMcfd (millions of cubic feet per day)	41667
	MMcfh (millions of cubic feet per hour)	1000000
	MMm3/d, **MMm3d** (millions of cubic metres per day)	1471250
	TJ/d (trillions of Joules per day)	39525000
	TJ/h (trillions of Joules per hour)	948600000
Heating Value Base Units = **Btu/cf**	**Btu/cf** (British Thermal Units per cubic foot)	1
	Btu/m3 (British Thermal Units per cubic metre)	0.028321
	Joules/cf (Joules per cubic foot)	0.00094805
	Joules/m3 (Joules per cubic metre)	0.000026849
	KJoules/cf, **KJ/cf** (thousands of Joules per cubic foot)	0.94805
	KJoules/m3, **KJ/m3** (thousands of Joules per cubic metre)	0.026849
	MBtu/m3 (thousands of British Thermal Units per cubic metre)	0.00002832
	MJoules/m3, **MJ/m3** (thousands of Joules per cubic metre)	26.849
Length (Pipe) Base Units = **Feet**	**Centimetre, cm**	0.03281
	Feet, ft	1
	Inches, in	1/12
	Kilometres, km	3281
	Metres, m	3.281
	Miles, mi	5280

Unit Group	*Supported Types*	*Conversion Factor*
Pressure, Pressure Drop Base Units = **Psi**	**Atmosphere**, **atm**	14.7
	Bar	14.504
	Feet H2O, **ft-wc**	0.4335
	Inches H20, **in-wc** (inches of water column)	1/27.707
	Inches HG, **in-HG** (inches of mercury)	0.491154
	Kg/cm2 (kilograms per square centimetres)	14.22
	Kg/m2 (kilograms per square metres)	0.001422
	KPa (thousands of Pascals)	0.14504
	Millibar, **mBar**	0.014504
	mm-H20 (millimetres of water column)	0.0014209
	mm-HG (millimetres of mercury)	0.0193367
	MPa (millions of Pascals)	14504
	Osi (ounces per square inch)	0.0625
	Pascals, **pa**	0.00014504
	Psi (pounds per square inch)	1
Temperature Base Units = **Rankine**	**Celsius**, **C**	$°C * 1.8 + 491.67$
	Fahrenheit, **F**	$°F + 459.67$
	Kelvin, **K**	$°K * 1.8$
	Rankine, **R**	1
Velocity Base Units = **Feet/sec**	**Feet/sec**, **Ft/sec**, **Fps** (feet per second)	1
	KMetres/hr, **KMh** (kilometres per hour)	0.91139
	Metres/sec, **Mps** (metres per second)	3.281
	Miles/hr, **Mph** (miles per hour)	1.4667
Viscosity Base Units = **Lbm/Ft-sec**	**Centipoise**, **cpoise**, **cp**	0.000672
	Lbf-sec/Ft2 (pound force second per square foot)	32.2

Unit Group	Supported Types	Conversion Factor
	Lbm/Ft-sec (pound mass per foot second)	1
	Micropoise	6.72 E-08
	Pascal-sec, **pa-sec** (Pascals per second)	0.672

Temperature Calculations

GASWorkS uses several equations to support its various temperature (thermal) calculations. These support equations are described in the following sections.

Average Temperature

Often, computations made by GASWorkS require determining the *Average Temperature* value along a pipe segment. When required, the average temperature is calculated as follows...

$$T_{AVE} = \frac{\left(T_1 + T_2\right)}{2}$$

Reference: None - Derived

Equation Variables

The following base units and variable definitions apply to this equation:

T_1 = Pipe Inlet (Upstream) Temperature or First Specified Temperature, Rankine
T_2 = Pipe Outlet (Downstream) Temperature or Second Specified Temperature, Rankine
T_{AVE} = Average Temperature, Rankine

Outlet Temperature - Compressor

The Outlet Temperature of a Compressor is estimated as follows...

$$T_2 = T_1 \times R^{\frac{k-1}{k}}$$

Where,

$$R = \frac{P_2}{P_1}$$

Reference 1

Equation Variables

The following base units and variable definitions apply to this equation:

k = Specific Heat Ratio, dimensionless
P_1 = Pipe Inlet (Upstream) Pressure, psia
P_2 = Pipe Outlet (Downstream) Pressure, psia
R = Compression Ratio, dimensionless
T_1 = Inlet Gas Flowing Temperature, Rankine
T_2 = Outlet Gas Flowing Temperature, Rankine

$P_1 = P_{1\_GAUGE} + P_{1\_ATM}$
P_{1_ATM} = Atmospheric Pressure at First Conditions, psia
P_{1_GAUGE} = Gauge Pressure at First Conditions, psig

$P_2 = P_{2\_GAUGE} + P_{2\_ATM}$
P_{2_ATM} = Atmospheric Pressure at Second Conditions, psia
P_{2_GAUGE} = Gauge Pressure at Second Conditions, psig

GASWorkS™ 10.1 *Appendix*

Joule-Thomson Temperature Drop

The temperature drop across regulators and along pipe segments is estimated by the following relationship. Note that this is an approximate equation and is only valid for natural gas mixtures composed largely of methane.

$$\Delta T = \frac{P_1 - P_2}{P_{ATM}} \times 1° \ Fahrenheit$$

Reference 1

Equation Variables

The following base units and variable definitions apply to this equation:

P_1 = Inlet Pressure, psia
P_2 = Outlet Pressure, psia
P_{ATM} = Average Atmospheric Pressure, psia
ΔT = Temperature Change, Fahrenheit

Pipe Temperature Change - *Overall Transfer Coefficient*

The outlet temperature of the gas flowing through a pipe segment based on an overall heat transfer coefficient is calculated as follows. This method predicts the outlet temperature of a pipe segment by using an overall heat transfer coefficient between the gas and the environment.

$$T_2 = T_{Amb} + (T_1 - T_W) \times e^{-\theta}$$

Where,

$$\theta = \frac{\pi DUL}{12\rho QC_P}$$

Reference 2

Equation Variables

The following base units and variable definitions apply to this equation:

C_P = Constant Pressure Specific Heat, dimensionless
D = Inside Diameter, inches
e = 2.71828, Napierian Constant
L = Physical Length, feet
π = 3.141592, Pi
Q = Volumetric Flow Rate, ft³/hr
ρ = Average Gas Density, lbm/Ft³
T_1 = Inlet Temperature, Fahrenheit
T_2 = Outlet Temperature, Fahrenheit
T_{Amb} = Ambient Temperature, Fahrenheit
θ = As defined in the equation definition
T_W = Pipe Wall Temperature, Fahrenheit
U = Overall Heat Transfer Coefficient, btu/(hr-ft²f)

Pipe Temperature Change - *Convection*

The outlet temperature of the gas flowing through a pipe segment based on convection (film) heat transfer is calculated using the following equation. The method was derived from various sources, including the noted references. This method predicts the outlet temperature of a pipe segment by assuming that the heat transfer occurs by convection between the flowing gas and the inner pipe wall. The pipe wall temperature is assumed to remain constant along the length of the pipe segment.

$$T_2 = T_W + (T_1 - T_W) \times e^{-\theta}$$

Where,

$$\theta = \frac{\pi DUL}{12 \rho Q C_P}$$

$$U = h_{film}$$

$$h_{film} = \frac{16.6 C_P (\rho V)^{0.8}}{D^{0.2}}$$

Reference 3, 4

Equation Variables

The following base units and variable definitions apply to this equation:

C_P = Constant Pressure Specific Heat, dimensionless
D = Inside Diameter, inches
e = 2.71828, Napierian Constant
h_{film} = Film Transfer Coefficient, dimensionless
L = Physical Length, feet
Q = Volumetric Flow Rate, ft³/hr
ρ = Average Gas Density, lbm/ft³
π = 3.141592, Pi
T_1 = Inlet Temperature, Fahrenheit
T_2 = Outlet Temperature, Fahrenheit
T_W = Pipe Wall Temperature, Fahrenheit
U = Overall Heat Transfer Coefficient, btu/(hr-ft²-f)
V = Average Gas Velocity, ft/sec

Pipe Temperature Change - *Convection & Conduction*

The outlet temperature of the gas flowing through a pipe segment based on both convection (film) and conduction heat transfer is calculated using the following equation. The method was derived from various sources, including the noted references. This method predicts the outlet temperature of a pipe segment by assuming that the heat transfer occurs by convection between the flowing gas and the pipe wall and by conduction through the pipe wall, insulation, and environment surrounding the pipe and insulation. The environment is only assumed to conduct to the extent of the influence zone. The influence zone is assumed to radially extend from the axis of the pipe outward to the edge of the influence zone. The size of the influence zone is computed as the product of the Influence Factor times the Outside Diameter. The "environment" represents the environment surrounding the pipe - soil, air, or liquid.

$$T_2 = T_S + (T_1 - T_S) \times e^{-\theta}$$

Where,

$$\theta = \frac{2\pi D U L}{\rho Q C_P}$$

$$\frac{1}{U} = \frac{1}{h_{film}} + \frac{D}{24}\left(\frac{\ln\left(\frac{D_O}{D}\right)}{K_P} + \frac{\ln\left(\frac{D_I}{D_O}\right)}{K_I} + \frac{\ln\left(\frac{D_S}{D_I}\right)}{K_S} \right)$$

$$h_{film} = \frac{16.6 C_P (\rho V)^{0.8}}{D^{0.2}}$$

$$D_I = D_O + t_I, \ Inches$$

$$D_S = D_O X_S, \ Inches$$

Reference 5

Equation Variables

The following base units and variable definitions apply to this equation:

C_P = Constant Pressure Specific Heat, dimensionless
D = Inside Diameter, inches
D_I = Pipe and Insulation Diameter, inches
D_O = Outside Pipe Diameter, inches
D_S = Pipe, Insulation, and Heated Soil Diameter, inches
e = 2.71828, Napierian Constant
h_{film} = Film Transfer Coefficient (See Convection Only method)

K_I = Thermal Conductivity of Insulation Material, btu/(hr-ft-f)
K_P = Thermal Conductivity of Pipe Wall Material, btu/(hr-ft-f)
K_S = Thermal Conductivity of Soil surrounding Pipe, btu/(hr-ft-f)
L = Physical Length, feet
Q = Volumetric Flow Rate, ft^3/hr
ρ = Average Gas Density, lbm/ft^3
π = 3.141592, Pi
t_I = Thickness of Insulation, inches
T_1 = Inlet Temperature, Fahrenheit
T_2 = Outlet Temperature, Fahrenheit
T_S = Temperature of the Soil at the edge of the Influence Zone, Fahrenheit
U = Overall Heat Transfer Coefficient, btu/(hr-ft^2-f)
V = Average Gas Velocity, ft/sec
X_S = Influence Factor

References

1) Gulf Publishing Company, *Pipe Line Rules of Thumb Handbook*, McAllister, Fourth Edition, 1998.
2) Gas Processors Suppliers Association, *Engineering Data Book*, Eleventh Edition - FPS, 1998.
3) Industrial Press, *Gas Engineers Handbook*, 1965.
4) CRC Press - Taylor & Francis, *Gas Pipeline Hydraulics*, 2005.
5) Pipeline And Gas Journal, May, 1974.

Valve Flow Equations

GASWorkS supports the calculation of the various values associated with the flow through a valve using a variety of industry and manufacturer equations. The various equations supported by GASWorkS are described in the following sections.

Crane

The Crane Valve flow equation used by GASWorkS is described as follows...

$$Q = 40656D^2 \times \sqrt{\frac{P_1(P_1 - P_2)}{F_k \times SG \times T_f}} \times E_v$$

Where,
$$F_k = Resistance\ Coefficient$$

And F_k Is Related To C_v As Follows,
$$F_k = \frac{894D^4}{C_v^2}$$
$$C_v = Published\ Valve\ Coefficient$$

Reference 1

Rockwell

The Rockwell Valve flow equation used by GASWorkS is described as follows...

$$Q = 40666D^2 \times \sqrt{\frac{P_1(P_1 - P_2)}{F \times SG \times Z \times T_f}} \times E_v$$

Where,
$$F = Pressure\ Drop\ Factor$$

Reference 2

GASWorkS™ 10.1 — *Appendix*

Equation Variables

Unless noted otherwise, the following base units and variable definitions apply to all of the methods and equations described in the *Valve Flow Equations* section:

D = Equivalent Inside Diameter Of Valve Body, inches
E_v = Throttling Factor, dimensionless
P_1 = Pipe Inlet (Upstream) Pressure, psia
P_2 = Pipe Outlet (Downstream) Pressure, psia
Q = Volumetric Flow Rate at the specified Base Pressure and Temperature, cfh
SG = Specific Gravity, dimensionless
T_f = Average Gas Flowing Temperature, Rankine
Z = Compressibility Factor, dimensionless

$P_1 = P_{1\_GAUGE} + P_{1\_ATM}$
P_{1_ATM} = Atmospheric Pressure at First Conditions, psia
P_{1_GAUGE} = Gauge Pressure at First Conditions, psig

$P_2 = P_{2\_GAUGE} + P_{2\_ATM}$
P_{2_ATM} = Atmospheric Pressure at Second Conditions, psia
P_{2_GAUGE} = Gauge Pressure at Second Conditions, psig

Throttling Factor

The following table summarizes the values of the Throttling Factor based on the specified Percent Open value. This relationship only applies to venturi-type plug valves; however, in the absence of information for other valve types, GASWorkS uses it as an approximate value for all valve types.

Percent Open Value	*Throttling Factor Value, E_v*
100	1.0
90	1.0
80	0.55
70	0.33
60	0.21
50	0.12
40	0.06
30	0.04

Percent Open Value	Throttling Factor Value, E_v
20	0.03
10	0.02
0	0
Reference 2	

References

1) Crane Technical Paper No. 410, *Flow of Fluids Through Valves, Fittings, and Pipe*.
2) Rockwell Publication V-117-R3, *Flow Characteristics of Rockwell Plug Valves.*

Well Flow Equation

As implemented by GASWorkS, the well flow calculation uses the following equation. The equation is sometimes known as the "top hole curve" equation. The Performance Coefficient and Equation Exponent parameters are calculated from a multi-point fit of the well's performance curve.

$$Q = C \times N \times \left(P_1^2 - P_2^2\right)^n \times E$$

Reference: Industrial Press, *Gas Engineers Handbook*, 1965.

Equation Variables

Unless otherwise noted, the following base units and variable definitions apply to all of the methods and equations described in the *Well Flow Equation* section:

    C = Performance Coefficient
    E = Performance Factor
    n = Equation Exponent
    N = Number of Wells Represented
    P_1 = Shut-in or Static Top Hole Pressure, psia
    P_2 = Flowing Top Hole Pressure, psia
    Q = Top Hole Volumetric Flow Rate at the specified Base Pressure and Temperature, cfh

GLOSSARY

Glossary

The following table describes some terms associated with the GASWorkS modeling environment.

Term	*Definition*
2-Point Pipe	A single-segment pipe. The pipe extends directly between the From Node and To Node locations.
Arc Pipe	A curved pipe segment.
Attribute Data	Refers to non-model data associated with customer, node, and pipe data. Non-model data values do not affect the calculation of the pressure and flow values in the system.
Balanced	Refers to a condition where all node errors are within the specified Convergence Tolerance value.
Base Pressure	Refers to the pressure used as the basis for stating a load or flow rate. When used with a customer load, the base pressure usually refers to the billing pressure.
Base Temperature	Refers to the temperature used as the basis for stating a load or flow rate. When used with a customer load, the base temperature usually refers to the billing temperature.
Branch Customer	A customer supplied from another customer's service line. A branch customer is not attached to a main.
Convergence Tolerance	A setting used to indicate when the solution is "close enough."
Customer Data File	A data file that contains the information associated with the customer features in a model.
Diameter	Refers to the Size/Type Code or inside diameter of a pipe segment.
Favorite	A User-selected Result.
From Node	The beginning node of a pipe feature.
GDI	An acronym (abbreviation) for the GASWorkS Graphic Data Interface.
Grips	A symbol displayed when a feature is selected. When clicked, the feature can be moved by dragging the grip symbol.
Group	A collection of User-selected features that may be moved, copied, or deleted as one.

Term	*Definition*
Header Data	The common data for the model such as base pressure and base temperature, and solution control values.
Hydraulic Type	Specifies the hydraulic element type, such as compressor, fitting, pipe, regulator, or valve.
Isometric View	A display method where three-dimensional objects are represented in classic isometric X, Y, and Z axis. Often used to display a model using 3D coordinate values.
Load	A flow leaving the system at a node. Opposite of a supply. A negative load value.
Model	The header, node, pipe, and customer data that defines the system being modeled.
Model Schematic	A line representation of the system being modeled.
Node	The end of a pipe or connection between multiple pipe ends.
Node Data File	A data file that contains the information associated with the nodes in a model.
Pipe	A model feature that serves as a graphic representation of a gas pipeline - typically a main, but can also be used for a service. Data associated with a pipe is used to calculate a steady-state solution to the model. A pipe also connects two nodes.
Pipe Data File	A data file that contains the information associated with the pipe features in a model.
Plan View	A display method depiction of image in the X-Y plane from the positive Z perspective. Often used to display models using 2D coordinate values.
Polyline Pipe	A multi-segment pipe. The end of each intermediate segment is referred to as a vertex.
Property Table	A set of data files which contain common data associated with the various pipes, devices, and fittings used by GASWorkS. Each Property Table contains specific information about the type it represents. For instance, the Pipe Property Table contains information on the pipe-type features, such as the *Inside Diameter* value.
Size/Type Code	A GASWorkS abbreviation used to identify a "pipe" feature's size, material, and/or distinguishing value. Used to establish the link between the pipe data files and the Property Tables.

Term	Definition
Supply	A flow entering the system at a node. Opposite of a load. A positive load value.
To Node	The ending node of a pipe feature.
Trunk Customer	A customer attached to a main, which then supplies another customer, known as a Branch Customer.
User Graphics	A graphic image, line, or symbol feature input and defined by the User.
User Table	A User-created spreadsheet or table-style graphic.
User Text	Text features input and defined by the User.
Valve Node	A node that is designated as a valve used by the Trace routines.

INDEX

GASWorkS™ 10.1

Index

GASWorkS™ 10.1

Index

GASWorkS™ 10.1

Index

GASWorkS™ 10.1 *Index*

GASWorkS™ 10.1

Index

694

GASWorkS™ 10.1 — *Index*

GASWorkS™ 10.1

Index