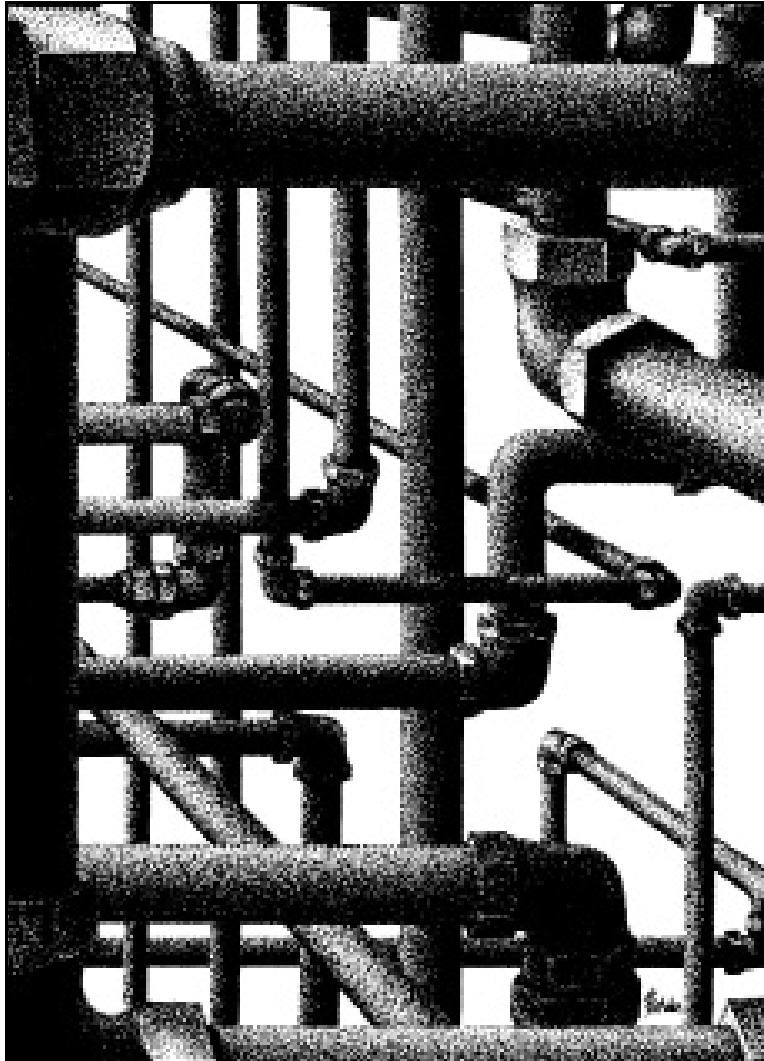


GASWorkS™ 11.0

Demonstration Guide



B3PE LLC
419 East Columbia Street
Colorado Springs, Colorado 80907 USA
(719) 578-9391
www.b3pellc.com

Thank you for your interest in our products and for taking the time to view our demonstration software. We hope you will be pleased with what you see during the demonstration. We are truly committed to providing products that ensure your success. If you have suggestions that might help improve GASWorkS to better meet your needs - *please* let us know your ideas. Again, thank you for your interest.

A handwritten signature in black ink, appearing to read 'B3' or 'BB', with a stylized flourish.

Bradley B. Bean, PE
Managing Member

**© 2025 B3PE LLC - All rights reserved.
Revision 001**

**GASWorkS and the “B-Cubed” logo are trademarks of B3PE LLC.
Windows and associated product names are a trademark of Microsoft Corporation.**

General Information	1
System/Hardware Requirements.....	1
Installing GASWorkS.....	1
Starting GASWorkS	2
Using The Help System	3
The Help Menu	3
Contact Us	3
Example	5
Creating A New Model	5
Setting The Default Data Values	7
Using The Graphic Data Interface (GDI) Window	9
Building The Model.....	10
Solve The Model	15
View The Analysis Results	16
Tap A Pipe	18
Color Coding Pipes & Nodes.....	19
Edit Pipe Data	21
Working With The Data Reports.....	22
What To Do Next.....	24
Uninstalling GASWorkS	25



General Information

The information contained in this guide is intended to provide an overview of the GASWorkS™ software by demonstrating a portion of its extensive feature set. Although the GASWorkS feature set is not fully demonstrated, we hope that the features used during the exercises will provide a good appreciation for how GASWorkS may be able to satisfy your network modeling needs.

The demonstration software is a full-featured, time-limited version of GASWorkS. Working the example in this guide should give you a good introduction to the software's features and ease of use. The example in this guide 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. The example exercise is a bit long; however, please be patient and persistent in working through it - completion of the example will provide a valuable lesson on how to use the software to efficiently meet your modeling needs. If you wish to explore the software further, we suggest reviewing the examples contained in the Help Guide.

System/Hardware Requirements

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

The GASWorkS software will require a minimum of 18 MB of free hard disk space when installed. An internet connection is required for the license management system, help documentation, and the use of the GeoMap functions.

GASWorkS uses the Microsoft .NET Framework for some functionality. An appropriate version (4.8 or newer) of the Framework must be installed on the machine for GASWorkS to function properly. Not all versions of Windows support the required version of the Framework.

Installing GASWorkS

An appropriate version of Windows must be running in order to install GASWorkS. If it is not, start Windows now, then proceed with the installation process described below.

Note...

You will need to be logged on as the Full Administrator or use the “Run as administrator” option when executing the installation file.

- Using the provided link, download the setup file (.exe) to your device. Using Windows File Explorer, navigate to the file's location.



- Double-click the setup file name. If a Windows Security warning appears, allow your device to execute the file.
- After the setup program has been started, several screens will be displayed during the installation process. Read each screen carefully and respond appropriately.

Note...

On the Terms Of Use screen, read the TERMS OF USE AGREEMENT carefully and select the “I agree to...” option. If you do not wish to accept the Terms Of Use, you will not be able to continue with the installation process.

- A message will be displayed when the installation process is complete. The message will indicate whether the installation was successful or failed. Select the appropriate option to clear the message.

Starting GASWorkS

Note...

- An appropriate version of Windows must be running before GASWorkS may be executed. If Windows is not running, start it now.
- The device must be connected to the internet the first time that the GASWorkS software is executed.
- You will need to be logged on as the Full Administrator or use the “Run as administrator” option the first time you execute the GASWorkS software.

- From the Windows Desktop, double-click the *GASWorkS 11 Demo* shortcut icon.

- The GASWorkS software will be started, and the Welcome screen will be displayed. Click the *Continue* command button.

- The Demonstration Notes screen will be displayed. Read carefully, then click the *Continue* command button. The Graphic Data Interface (GDI) Window will be displayed.



Using The Help System

The help documents are stored on the B3PE server. You must be connected to the internet to access the various GASWorkS help and support documents.

The GASWorkS Help Guide can be used to view examples and how-to tips on performing a specific task. The GASWorkS User's Manual provides information on specific screens and commands, as well as detailed instructions, descriptions, and explanations.

Once a document is opened, it can be saved locally if you wish. If you save a help document to your machine for offline access, you must open it directly in your Portable Document Format (PDF) document viewer. A suitable PDF document viewer or reader is required to view and print the documents.

Note...

GASWorkS first looks for help files in the “docs” folder of the “app” folder of the GASWorkS root directory. If the requested file exists, it will be opened from that location, eliminating the need to download it from our server. It is suggested that you save downloaded help files to that location (usually “c:\Program files (x86)\GASWorkS 11\app\docs”) for faster display.

The Help Menu

Access to the various help documents and information is provided in the *Help* menu. You can also view our contact information, send a help request, or visit our website. *Note* - Some features require an internet connection to work properly.

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 at any time - we may be in.

By Telephone - (719) 578-9391

By Email - help@b3pe.com

By Website - www.b3pellc.com





Example

Various example files are included when GASWorkS is installed on the hard disk. These files are installed in the “files” subdirectory of the GASWorkS root directory. The example files contain a variety of system types. You can view, manipulate, and solve these example files - or use the instructions in the following sections to create, edit, solve, and report your example model.

In this example, a model will be developed for a simple piping system representing a light industrial park using generic loads. Once the model is built and solved, a new connection and demand will be added to the system. The new load will exceed the system’s capacity. Changes will be made to the system to allow the addition of the load. During the example, you will learn how to create a new model using various coordinate entry methods, solve a model, display graphic results, color code based on result values, edit pipe and node data, tap an existing pipe, and use the report features.

The resulting model should look similar to the image in Figure 2 (page 14).

Note...

- GASWorkS has an automatic save feature which can be turned “on” or “off” depending on the User’s preference. As initially installed, the option is turned “on.” To change this preference option, select the *Options & Settings* item from the *Utilities* menu. Select the *Environment* data tab from the User Options & Settings screen.
- This example assumes that the settings, options, and default data values are set to the originally installed values. If they have been changed, some of the example steps may not behave or appear as described.

If GASWorkS is not running, start it now. If a model is already open, close it now by selecting the *Close* item from the *File* menu.

Creating A New Model

The first step in creating a model is to open a new set of model files. To open a new model...

- From the *File* menu, select the *New* item. The Save screen will be displayed.
 - In the *File name* data field, enter a name for the new model. For example, type **MyDemo** in the data field.
 - From the *Save as type* list, select **GASWorkS 11 (*.gwz)**.



- When an appropriate New Model Name has been entered and the type has been selected, click the *Save* command button to proceed.

Note...

GASWorkS 11.0 supports two types of model file formats - the traditional or legacy “hdr” file format (GASWorkS 10.2 and older) - and the GASWorkS 11 “gwz” format. The legacy format stores the model as a group of individual files. The new format stores the model as a single file.

The Graphic Data Interface (GDI) Window should be displayed.

Note...

- GASWorkS 11 supports two types of graphic data interfaces: the classic or legacy “toolbar” type interface, and the new GASWorkS 11 interface, which replaces the toolbars with the Command Tree. To use the legacy interface, select the *Use Classic GDI Format* option from the *Environment* data tab of the User Options & Settings screen.
- The instructions provided in this document presume that the “new” GDI format will be used.

Note - If the GDI Window is not automatically displayed, select the *View/Edit* item from the *Graphics* menu to display the GDI Window.

If the GDI Window is not sized to fill the GASWorkS application screen, resize the GDI Window using the *Maximize GDI Window* command located either in the GDI Command List or under the *GDI Utilities* branch of the GDI Command Tree.

Note...

- To assist in building a model using the drawing method, GASWorkS provides two “snap” methods that cause a pipe end to be snapped to an existing node. The first snap mode is called “auto-snap.” This is implemented by holding down the *Shift* key and left-clicking the mouse when prompted for a node location.
- The second method is called Feature Snap; this method snaps to an existing feature (node) found within a certain distance of an entered location. To use this method, click the *Feature Snap* icon from the lower-left corner of the GDI Window until the icon’s background is darkened. When this method is turned “on,” a circle or target will be displayed around the mouse crosshairs when prompted for a location. This circle represents the snap distance. When a feature, for example, a node, is located within the target and the left mouse button is clicked, the location of the targeted feature will be used as the selection location.

For this example, ensure that the Feature Snap is turned “on.” Click on the *Feature Snap* icon located in the lower-left corner of the GDI Window until the icon’s background is darkened. The Feature Snap will automatically force (snap) the new coordinate selection to an existing feature within the Snap Target of the selected location.



Setting The Default Data Values

When creating a new model or adding new features to an existing one, the data values for the new features are set to the current default data values. If similar models are being created, these values only need to be set once. However, if models that are not similar are being created, the values must be appropriately set when changing model types.

An alternative to using the default data to populate values for newly entered features is to use the data from the last similar feature in the model to populate the values for a new feature. To use this method, select the *Use Last Record As Default* option on the *Default Data* tab on the User Options & Settings screen.

Note...

In this document, two values are listed where a data value is given. The first value represents the value in US (Imperial) dimensional units. The second value (enclosed in brackets []) represents the value in UK (Metric) dimensional units. Use the appropriate values.

To set the default data values...

- From the *Utilities* menu, select the *Options & Settings* item. The User Options & Settings screen will be displayed.

- On the *Units* data tab, set the following values:

Coordinate Values = **Feet** [Metres]
Customer Loads = **Mcfh** [M3h]
Diameter Values = **Inches** [Millimetre]
Efficiency Values = **Decimal**
Elevation Values = **Feet** [Metres]
Heating Value (Energy Content) Values = **Btu/cf** [MJoules/m3]
Length Values = **Feet** [Metres]
Node Load Values = **Mcfh** [M3h]
Pipe Flow Values = **Mcfh** [M3h]
Pressure Values = **Psi** [mBar]
Prompted Length Values = **Feet** [Metres]
Temperature Values = **Fahrenheit** [Celsius]
Velocity Values = **Feet/sec** [Metres/sec]
Viscosity Values = **Lbm/Ft-sec** [Centipoise]

- On the *Default Data* tab, set the following values:

Use Last Record As Default = **Unselected**



Customer Load = **0 Mcfh** [0 M3h]
Node Base Load = **-10 Mcfh** [-280 M3h]
Node Base Load Known = **Yes**
Node Elevation = **0 Feet** [0 Metres]
Node Pressure = **0 Psi** [0 mBar]
Node Pressure Known = **No**
Node (Gas) Temperature = **60 Fahrenheit** [15.6 Celsius]
Node Temperature Known = **Yes**
Pipe Flow Equation = **IGT-Improved**
Pipe Hydraulic Efficiency = **0.95 Decimal**
Pipe Size/Type (Inside Diameter) = **2P Inches** [63P-11 Millimetre]
Allow Pipe Sizing = **No**

Miscellaneous Data Values...

Pipe Facility Type = **Unspecified**
Node Increment = **5**
Pipe Sizing Group = **Any**

- On the *Gas Properties* data tab, set the following values:

Base Pressure = **14.73 Psi** [1013.25 Millibar]
Base Temperature = **60 Fahrenheit** [15.6 Celsius]

Specific Gravity = **0.6**
Viscosity = **0.000007 Lbm/Ft-sec** [0.0104 Centipoise]
Heating Value = **1000 Btu/cf** [37.2 MJoules/m3]
Specific Heat Ratio = **1.3**
Values Are Unknown = **Unselected**

- On the *Graphics* data tab, set the following values:

Flow Arrow Size = **3 % Of Display Width**

- On the *Reports* data tab, set the following values:

Negative Pressure Nodes = **Selected**

- On the *Environment* data tab, set the following values:

Allow Undo Of Data/Graphic Changes = **Selected**
Always Maximize GDI Window = **Selected**
Automatically Open GDI Window = **Selected**
Automatically Save At Timed Increments = **Selected**

Display Tooltips = **Selected**

Prompt For Data During New Feature Entry = **Selected**

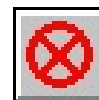
Note - Select the remaining options as desired.

- The remaining values and settings should be appropriate. Click the *Apply* command button to apply the settings. Then, click the *Close* command button to close the screen.

Using The Graphic Data Interface (GDI) Window

When using GASWorkS, much of your time will be spent working within the GDI Window, so let's take some time to describe its various features. Commonly used commands are listed in the Command Tree located on the upper-left side of the GDI Window. The tree contains various commands grouped by functionality. Select a branch to expand the group, then select the item associated with the desired command.

In the GDI Window, a GDI Command is executed either by clicking the appropriate GDI Icon, by selecting a command from the GDI Command Tree, by selecting an item from the GDI Command List, or by typing the command name or abbreviation on the GDI Command Line. In this example, we will use a variety of execution methods. If a GDI Command is accidentally selected, it may be canceled by clicking the *Cancel* icon.



The GDI Command List icon is located below the GDI Command Tree on the left side of the GDI Window. Select a command from the list to execute.

The GDI Command Line is located at the bottom of the GDI Window, below where the mouse coordinate values are displayed. Type a command name or abbreviation in the command line and press the *Enter* key to execute the command.

When a feature is selected or a new feature is added, the associated data will be displayed in the upper-right side of the GDI Window in the Data Panel. If changes are made to the associated feature, click the *Apply Data Values* command button to save and apply those changes.

The main portion of the GDI Window is comprised of the GDI Display. The GDI Display is the portion of the screen where the model image (GDI Image) is displayed. The *Scroll* icon is located in the lower-left corner of the GDI Window. The view can be moved right or left and up or down by clicking one of the arrow symbols on the icon. The *Zoom To Fit*, *Zoom In*, *Zoom Out*, and *Zoom Previous* icons are located next to the *Scroll* icon. There are various other methods for zooming and panning, including using the mouse wheel and buttons.

When a GDI Command requires keyboard input, a prompt will appear just above the GDI Display.



Note...

- GASWorkS provides several ways to enter a node's location. These include:

Specifying an X-Y coordinate pair;
Specifying a relative distance and angle;
Drawing the segment with the mouse.

- We will use each of these methods when constructing the example model.

Note - If your Windows environment is set to use a comma “,” as the decimal separator, use a space to separate the coordinate values.

Building The Model

Add The First Pipe Using The Coordinate Method Of Entry

- Click the *Add 2 Point Pipe* item from the *Construction* branch of the GDI Command Tree.
 - For the *From Node Location*, type **1000,1000** on the GDI Prompt Line and press the *Enter* key.
 - The Node Data will be displayed in the Data Panel. In the *Hydraulic Data Items* section, set the following values:

Name = **SUPPLY**
Pressure = **20** [2000]
Pressure Units = **Psi** [mBar]
Pressure Known = **Yes**
Note - The *Base Load Known* should automatically be set to “No.”
 - The remaining values should be acceptable. Click the *Apply Data Values* command button located at the bottom of the Data Panel to continue.

Note...

By changing the name of the node to “SUPPLY” does not make the node a supply point; setting the pressure as “Known” and the flow as “Unknown” causes this location to be a supply point. Nodes can have most any name up to 40 alphanumeric characters.



Note...

The setting of the “*Known*” items indicates whether the associated value is known or unknown. When a “*Known*” item is set to “No,” the value is unknown and will be calculated during the Solution routine. At this node, we indicated that the pressure was known (set to “Yes”) and that the base load was unknown (set to “No”). Generally, either the pressure value or the load value must be “Known” at each node, but not both. At supply nodes, the pressure is usually set as “Known” and the load as “Unknown.” At demand nodes, the pressure is usually set as “Unknown,” and the load is set as “Known.” A negative load value indicates that gas is leaving the system.

Note...

Depending on the *Zoom Scale*, a node symbol may appear at the new node’s location. Whether a node symbol is displayed or not, a “stretch” line should be displayed from the entered node location. We will zoom in later to make the view easier to see.

- For the *To Node Location*, type **1500,1000** [1150,1000] on the GDI Prompt Line and press the *Enter* key.
- The data for the new node will be displayed in the Data Panel. The displayed values should be acceptable. Either click the *Continue* command button or right-click the mouse to continue.
- The data for the new pipe will be displayed in the Data Panel. In the *Hydraulic Data Items* section, set the following values:

Size/Type Code = **4P** [125PE-S11]
Diameter Units = **Inches** [Millimetre]

- Click the *Apply Data Values* command button to continue.

Note...

The *Size/Type Code* list contains various predefined Size/Type Codes. Each code represents an entry in the Pipe Property Table. GASWorkS uses Property Tables to store redundant data items, such as *Inside Diameter* values. Property Tables are provided for each hydraulic feature type. For example, pipes, fittings, regulators, valves, compressors, and well features. The User may add to or revise the data contained in any of the Property Tables.

To make the pipe we just entered easier to see, let’s zoom in on the GDI Display. Click the *Zoom To Fit* icon in the lower-left corner of the GDI Window to zoom the GDI Image to fit the GDI Display. A single horizontal pipe with a node on each end should be displayed.



Add The Second Pipe Using The Relative Distance & Angle Method

- Click the *Add 2 Point Pipe* item from the *Construction* branch of the GDI Command Tree.
 - For the *From Node Location*, right-click the mouse to accept the default From Node coordinates (1500,1000 [1150,1000]).

Note...

GASWorkS remembers the coordinates of the last node that was entered. When a default coordinate pair is displayed, either press the *Enter* key or right-click the mouse to accept the coordinates.

- For the *To Node Location*, type @500<180 [@150<180] on the GDI Prompt Line and press the *Enter* key.

Note...

The “@” symbol tells GASWorkS that a relative distance and angle will be used to locate the node. The node location is calculated relative to the From Node Location. Angles are measured clockwise from the 12 o’clock position (top of display) - zero (0) represents north.

- The data for the new node will be displayed in the Data Panel. The displayed values should be acceptable. Either click the *Continue* command button or right-click the mouse to continue.

Note...

Notice that the Node Data did not display for the From Node, but it did display for the To Node. That is because the data is only displayed when a new feature is entered. The From Node was previously entered, so the data was not displayed.

- The data for the new pipe will be displayed in the Data Panel. In the *Hydraulic Data Items* section, set the following values:

Size/Type Code = **2P** [125PE-S11]

- Click the *Apply Data Values* command button to continue.

To make the pipes we just entered easier to see, let’s zoom in on the GDI Display. Click the *Zoom To Fit* icon in the lower-left corner of the GDI Window to zoom the GDI Image to fit the GDI Display.



Add The Third Pipe By Using The Drawing Entry Method

- Select the *Add 2-Point Pipe* item from the GDI Command List.
 - For the *From Node Location*, move the mouse crosshairs so that **Node 15** is within the selection target. Left-click the mouse to indicate the node's location.

Note...

If the Node Data is displayed in the Data Panel, it indicates that the node's location was not properly selected. To reselect the From Node Location, click the *Cancel* icon and re-execute the *Add 2-Point Pipe* command.

- For the *To Node Location*, move the mouse crosshairs to a location below **Node SUPPLY** and to the left of **Node 15** (similar to **Node 20** in Figure 1). Left-click the mouse to indicate the node's location.
- The data for the new node will be displayed in the Data Panel. The displayed values should be acceptable. Either click the *Continue* command button or right-click the mouse to continue.
- The data for the new pipe will be displayed in the Data Panel. The displayed values should be acceptable. Either click the *Continue* command button or right-click the mouse to continue.

After entering this pipe, the model should look similar to Figure 1.

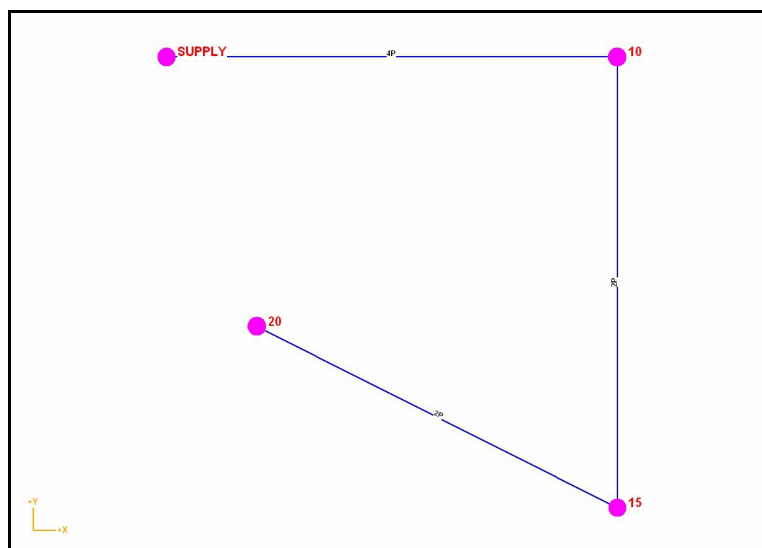


Figure 1

Add The Fourth (Last) Pipe

- Double right-click the mouse while the mouse pointer is in the GDI Display to re-execute the last GDI Command (*Add 2-Point Pipe* command).

- For the *From Node Location*, right-click the mouse to accept the default coordinates.
- For the *To Node Location*, type the letter **N**, a space, then followed by the node's name **SUPPLY** (**N SUPPLY**). Right-click the mouse, and press the *Enter* key to select the indicated node's location.

Note...

By including the "N" prefix, you are indicating that a node name will follow and to use that node's location for the To Node Location.

- The data for the new pipe will be displayed in the Data Panel. The displayed values should be acceptable. Either click the *Continue* command button or right-click the mouse to continue.

After entering the last pipe, the model should look similar to Figure 2.

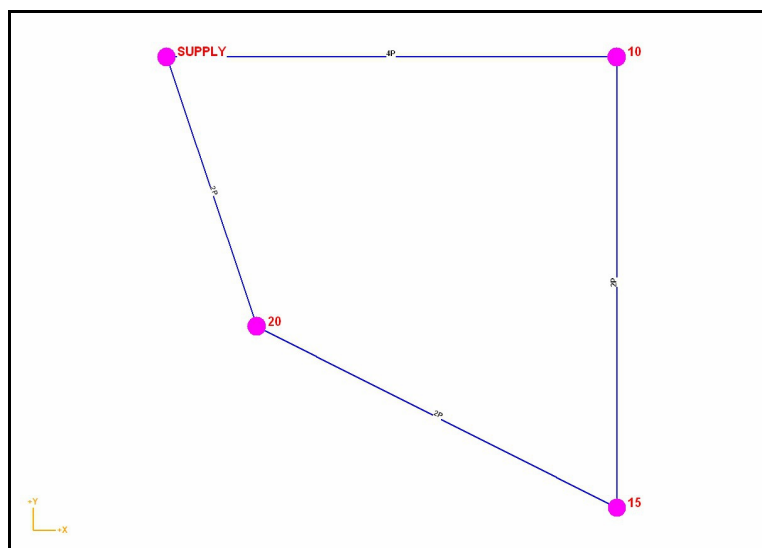
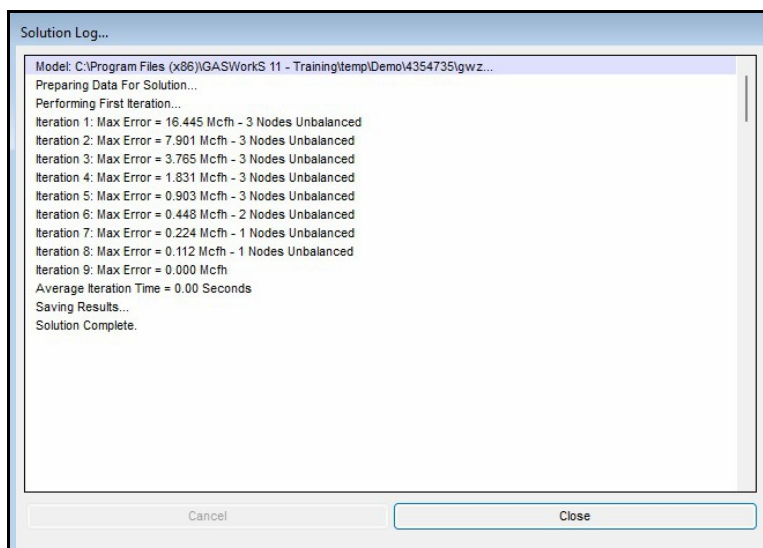


Figure 2

Solve The Model

Now that the model has been created, the unknown pressure and flow values may be calculated by “solving” the model. To solve the model...

- Click the *Solve* icon from the lower-right corner of the GDI Window. The Solution Data screen will be displayed.
- For this example, the default Solution parameters should be acceptable. Click the *Solve* command button to proceed with the Solution.
- While the Solution is performed, the Solution Log will be displayed. The Solution Log displays the Solution activity and informs the User when the Solution is complete.



Note...

- The Solution Log displays the progress of the Solution routine. In a normal solution, the node errors should start relatively large and decrease as the Solution progresses until the node errors are less than the specified convergence tolerance. The last line of the Solution Log should indicate “Solution Complete.” If the Solution is not complete, or exceptions are reported, recheck the model creation steps, correct any errors you encounter, then re-solve the model.
- The values displayed in your Solution Log will most likely not be the same as the values listed above. However, they should be close, and the last line should indicate “Solution Complete”. If the log does not say that the solution is complete. Retrace and check the model-building steps, then and try again.

- Upon completion of the solution, click the *Close* command button to close the Solution Log and return to the GDI Window.

When the solution is complete, the *Solve* icon symbol will change. The scale is now blue and balanced, indicating that the model is balanced.



View The Analysis Results

Now, let's review the analysis results. In addition to its various Report routines, GASWorkS provides several graphical methods for viewing the model data and the analysis results. We will look at the Report routines later; for now, let's use some graphical methods to view the analysis results.

Notice that the flow arrows are now displayed, indicating the direction of the pipe flow.

Note...

If the flow arrows aren't displayed, click the *Display Flow Arrows* item from the *Display Settings* branch of the Command Tree. Realize that sometimes the flow arrow symbols might be obscured by the pipe size text.

Let's display the pressure values for the nodes...

- Click the *Text Display Settings* item from the *Display Settings* branch of the GDI Command Tree. The Text Display Settings screen will be displayed.

- On the *Node Items* data tab, set the following items:

Display Node Text Items = **Selected**
Transparent Font = **Unselected**
Node Pressure = **Selected**

- Click the *Apply* command button to apply the changes to the display.

The pressure values and flow directions should be similar to those shown in Figure 3.

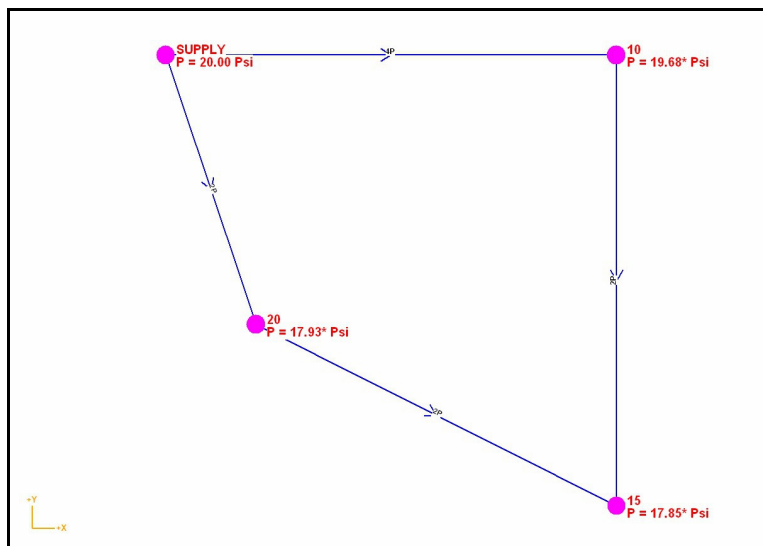


Figure 3

Note...

- The displayed pressure values represent the estimated pressure for each node point. As a result of the “drawing” of the third and fourth pipe segments, each example model is created slightly differently. The different pipe lengths will cause different pipe resistance, and the different pipe resistance will cause different pressure drop and pressure values. A large difference indicates an error has occurred during the model creation. If so, you might want to retrace the steps to identify the problem.
- The analysis results for an individual pipe or node can be viewed and printed using the *Edit Node* and *Edit Pipe* commands or by clicking the mouse on the desired feature.
- Sometimes, it is desired to move a text value to make it or another feature more visible or readable. An easy way to move text is to:
 - Click on the associated feature (node, pipe, customer). When selected, two boxes will appear. These are known as grips.
 - Click in the grip nearest the text value.
 - Move the mouse and text outline to the desired location.
 - Click the left mouse button.

Tap A Pipe

To simulate a new connection to the system, let's add a new node (a pipe tap) to the pipe segment between Node 15 and Node 20. To make a tap...

- Click the *Tap A Pipe* item from the *Edit Graphics* branch of the GDI Command Tree.
 - At the *Select Pipe To Tap* prompt, move the mouse pointer to the pipe between **Node 15** and **Node 20** and left-click the mouse. The selected pipe segment will be highlighted "yellow."
 - For the *Tap Location*, move the mouse crosshairs midway along the pipe segment between **Node 15** and **Node 20** and left-click the mouse. A node will be placed at the selected location.
 - The data for the new node will be displayed in the Data Panel. In the *Hydraulic Data Items* section, set the following values:

Name = **NEWTAP**
Base Load = **-40** [-2500]
Load Units = **Mcfh** [M3h]

- The remaining data values should be acceptable. Click the *Apply Data Values* command button to continue.

Note...

- The load for the Node NEWTAP was entered as a negative value. Remember that a negative value indicates a load leaving the system (a demand).
- Because a change has been made to the model, the previously computed pressure and flow values are no longer valid. To see the effect of the new load on the system, we will need to re-solve the model. Notice that the *Solve* icon has changed - the scale is now unbalanced, indicating that the model needs to be solved.



Re-solve the model by clicking the *Quick Solve* icon in the lower-right corner of the GDI Window. A message will appear warning of possible exceptions. Click the *Yes* command button to view the exceptions listed in the Solution Log. Note the locations of the listed features, or double-click on an exception row to flag and zoom to that location in the GDI. To redisplay the Solution Log, click on the *Solution Log Report* item in the *Window* menu list.



In Figure 4, the pressure values are negative (less than zero) at the noted locations. This indicates that the system has “run out of gas.” In other words, the increased demand caused by the new load is greater than the system’s capacity.

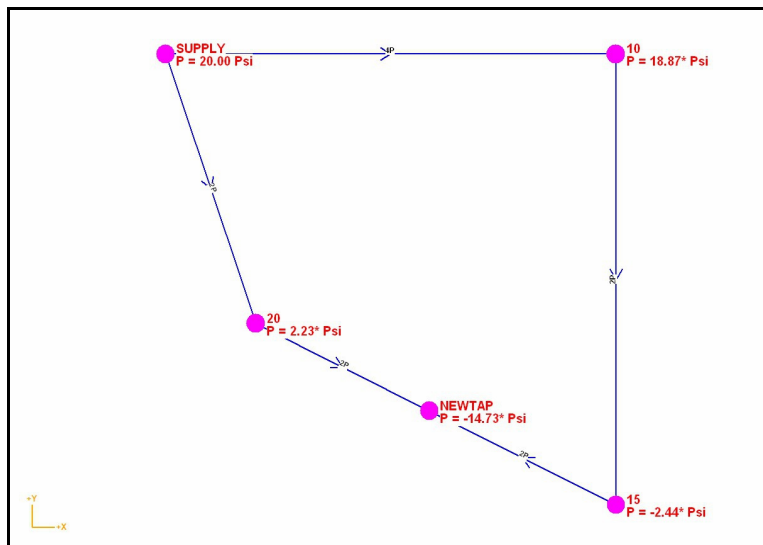


Figure 4

To correct this issue, normally, you might increase the size of the pipes with the highest pressure drop. Let’s use the color-coding features to identify the pipe or pipes with the highest pressure drop, and increase their size.

Color Coding Pipes & Nodes

GASWorkS provides several routines for color-coding pipes and nodes based on their data values. Features may be colored based on the results of a query. For example, pipes with pressures value below a specific pressure value could be colored red. Features may also be colored based on a range of values. The range values and colors may be set by the User or automatically set by GASWorkS. Let’s use one of the automatic range methods to color-code the pipes.

To determine which pipes have the highest pressure drop, we will color-code them based on their pressure drop values.

- Click the *Color Display Settings* item from the *Display Settings* branch of the GDI Command Tree. The Color Display Settings screen will be displayed.

- On the *Range Colors* data tab, set the following values:

Item = **Pipe Pressure Drop**
Units = **Psi [mBar]**

- Click the *Calculate* command button to automatically set the range values. Values will be displayed in the *From* and *To* data fields.
- Click the *Apply Range Colors* command button. The colors of the displayed pipes will be updated to reflect the range color values similar to those shown in Figure 5.

Note...

If the color settings are not to your liking, the User can select new color values by clicking in a color swatch to select a new color.

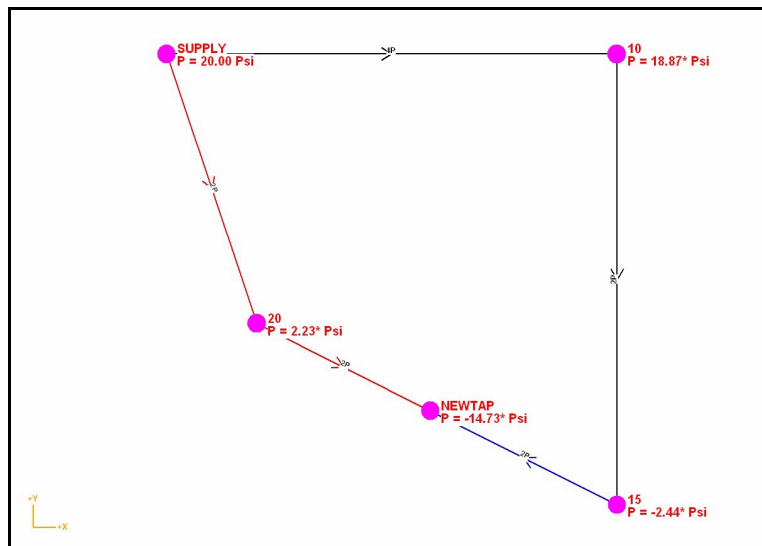


Figure 5

As a reminder of the range color values, the Color Legend will be displayed below the GDI Command List on the lower-left side of the GDI Window. The Color Legend will be displayed using the calculated color and value ranges.

Pipe Pressure Drop, Psi...	
	Below 3.653
	3.653 <= 6.176
	6.176 <= 8.698
	8.698 <= 11.220
	11.220 <= 13.743
	13.743 <= 16.265
	16.265 <= 18.787
	Above 18.787

Note...

In this case, the pipes with the highest pressure drop are colored “black.” The segments between Node 10 and Node 15 and between Node SUPPLY and Node 20 might be colored “black,” or one or the other might only be colored “black.” The results vary because of the varied pipe lengths resulting from the drawn pipe segments.

Let’s try to fix the pressure problem at Node NEWTAP by changing the size of the pipe between Node SUPPLY and Node 20.

Edit Pipe Data

During some previous steps, we used the GDI edit features to change the data associated with the new nodes and pipes we entered. Now, let’s use the GDI’s edit features to edit the size of the pipe between Node SUPPLY and Node 20.

- Select the pipe segment between **Node SUPPLY** and **Node 20** by moving the mouse pointer near the pipe segment and left-clicking the mouse.

- The data for the selected pipe will be displayed in the Data Panel. In the *Hydraulic Data Items* section, set the following values:

Size/Type Code = **4P** [125PE-S11]
Diameter Units = **Inches** [Millimetre]

- Click the *Apply Data Values* command button to save the changes.
- Click the *Solve* icon in the lower-right corner of the GDI Window. The Solution Data screen will be displayed.



- On the *Solution Data* tab, set the following values:

Update Color Coding = **Selected**

- Click the *Solve* command button. The Solution Log will be displayed. Review the results, then click the *Close* command button to close the log.

We can see that all of the node pressures are now positive. If the current values are adequate, we can conclude that the change fixed the problem; otherwise, we can change additional pipe sizes and re-solve the model to see the results.



Note...

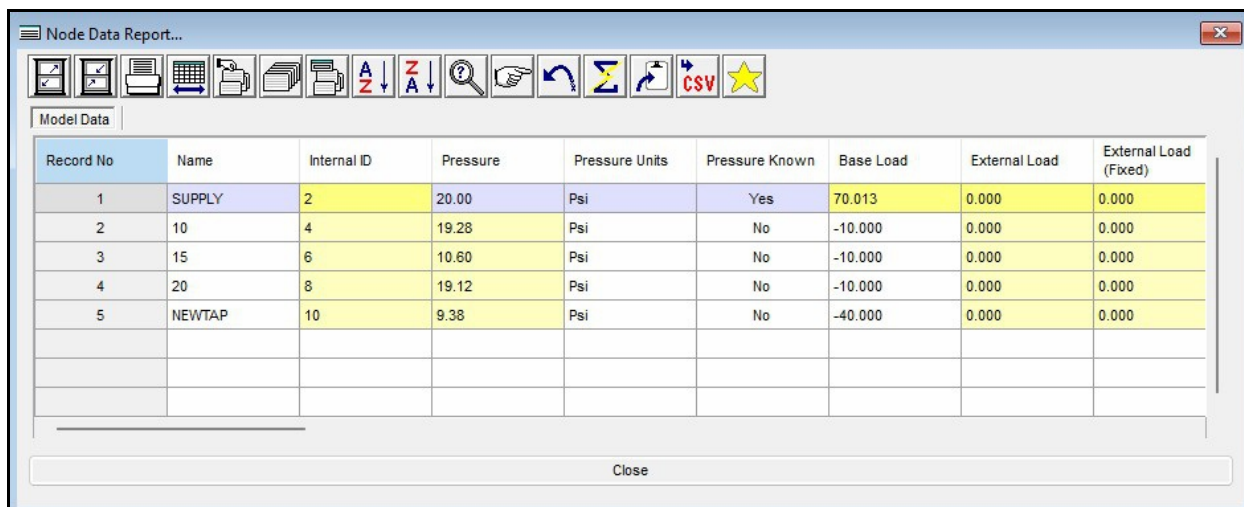
Notice that the pipe colors were updated to reflect the new pressure drop values. By selecting the *Update Color Coding* option, the last range color values were used to update the GDI feature colors after the solution was complete.

We can reset the feature colors (pipe colors) to their original color values. To do this, on the GDI Command Line, type **RESET** and press the *Enter* key. The feature colors (pipe colors) will now be displayed in the original default colors.

Working With The Data Reports

To view the Node Data Report...

- From the *Report* menu, select the *Node Data* item. The Node Data Report will be displayed.



The screenshot shows the 'Node Data Report...' window. It contains a table with the following data:

Record No	Name	Internal ID	Pressure	Pressure Units	Pressure Known	Base Load	External Load	External Load (Fixed)
1	SUPPLY	2	20.00	Psi	Yes	70.013	0.000	0.000
2	10	4	19.28	Psi	No	-10.000	0.000	0.000
3	15	6	10.60	Psi	No	-10.000	0.000	0.000
4	20	8	19.12	Psi	No	-10.000	0.000	0.000
5	NEWTAP	10	9.38	Psi	No	-40.000	0.000	0.000

At the bottom of the window is a 'Close' button.

Sort the results in the Node Data Report from the lowest to the highest pressure value. To do this...

- Click the *Make Selection Set* icon. The Query Specifications screen will be displayed. Set the following values:



Where = **Node Pressure**
 Is = **Min**
 To = **10**

- Click the *Perform Query* command button to create the selection set. A message will be displayed

stating the number of matches found. Click the *OK* command button to clear the message.

- Click the *Close* command button on the Query Specifications screen. The pressure node values should now be listed from lowest to highest.

Record No	Name	Internal ID	Pressure	Pressure Units	Pressure Known	Base Load
5	NEWTAP	10	9.38	Psi	No	-40.000
3	15	6	10.60	Psi	No	-10.000
4	20	8	19.12	Psi	No	-10.000
2	10	4	19.28	Psi	No	-10.000
1	SUPPLY	2	20.00	Psi	Yes	70.013

Note...

- The contents of the reports can be manipulated by creating “selection sets” based on User-specified selection criteria. The previous few steps created a sorted selection set, which found the ten (10) lowest pressure node values. In this case, only five nodes were in the model, so all records were included in the selection set, ranked from lowest to highest. Successive selection sets can be created.

- A selection set can be “cleared” by choosing the *Reset Selection Set* icon.



- Click the *Close* command button to close the Node Data Report.

From the *Report* menu, select the *Pipe Data* item. The Pipe Data Report will be displayed. Use the horizontal scroll bar to view items displayed on the right side of the report. As you scroll, you will see the various items reported for the pipe features, including the calculated Velocity, Volume, Flow Rate, and Pressure Drop values for each segment. When you are done browsing the report, click the *Close* command button.

Note...

- This example barely touched on the many features associated with the various reports. The reports allow data editing directly from the report screen by simply clicking on the appropriate data cell. The User may control the report’s contents by expanding and collapsing columns and creating query-based selection sets. Several routines are also provided for sorting and browsing through the report contents.
- To better fit the report on the printed page, try reducing the number of columns by using the Set Columns routine to specify which columns to display and print. When the *Set Columns* icon is clicked, the Column Settings screen will be displayed. The selected column names represent the columns that are currently displayed. Unselect some of the columns.
- The reports can be saved to the Windows Clipboard and pasted into another application, or saved as a Comma Separated Values (.csv) file and opened in a spreadsheet application.

What To Do Next

These last few steps demonstrate the most powerful use of GASWorkS: the ability to study the effects of system modifications before they are implemented. By analyzing these effects using the model, changes that might adversely affect the system may be identified before they become a real problem. Furthermore, solutions to counter or fix the results of a negative change can be easily tested in the model, avoiding installing incorrectly sized facilities in the field.

There are several pre-built example models distributed with the software. Refer to the Help Guide in the *Help* menu for descriptions and instructions for viewing the various example models. These pre-built examples include applications that use regulators, hydraulic valves, fittings, compressors, and wells, and demonstrate many of the software's features. Other examples can be found in the Training Examples section of our website (www.b3pelle.com).

Instructions for using the many commands are included in the Help Guide.

This simple example provided a brief overview of the basic routines found in the GASWorkS software. Many routines were not used in the example: the Import and Export routines, attribute valves, tracing, data checking, the use of the Pipe Sizing routine, the use of the customer data features, the use of a background image, and the use of the GeoMapping features were all absent from the demonstration example. Perhaps you may wish to explore some of these features on your own. These features are found in the User's Manual or the Help Guide. These documents may be downloaded from the *Support* section of our website.

When you are done exploring the software, select the *Exit GASWorkS* item from the *File* menu to close the program. A message may be displayed asking whether to save or discard the changes made to the model - select the appropriate option. If you choose not to save the changes, they will be ignored.

Uninstalling GASWorkS

If, after using the GASWorkS demonstration software, you wish to remove it, select the *Uninstall GASWorkS 11 Demo* item from the *GASWorkS Programs* folder.

- After the Uninstall Program has been started, read the contents of each screen carefully and respond appropriately.

Note...

If the Uninstall Program asks to remove any shared system files, click the *No* command button.

- A message will be displayed when the Uninstall routine is complete. The message will indicate whether the routine was successful or whether it failed. Select the appropriate option to clear the message.

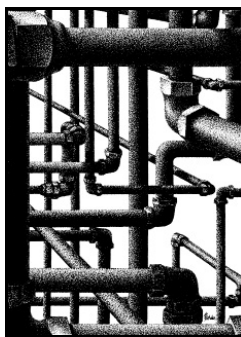
Note...

- On some devices, the Uninstall routine will not be able to remove any directory or file that has been created after the initial installation.

- Some of the files and/or folders created by GASWorkS after the installation may include files in the initialization directory (ini), files in the default model file directory (files), and/or files in the application directory (app).

- To remove these files and/or folders, use Windows File Explorer to find and delete the remaining files and folders. An example of the path for the initialization folder is shown below. The other folders have similar path locations.

- 32-bit machines - **C:\Program Files\GASWorkS 11 Demo\ini**
- 64-bit machines - **C:\Program Files (x86)\GASWorkS 11 Demo\ini**



B3PE LLC

Engineering & Software Services

Revision - 001, Copyright 2025, All Rights Reserved.