GASWorkS™ 10
Demonstration Guide
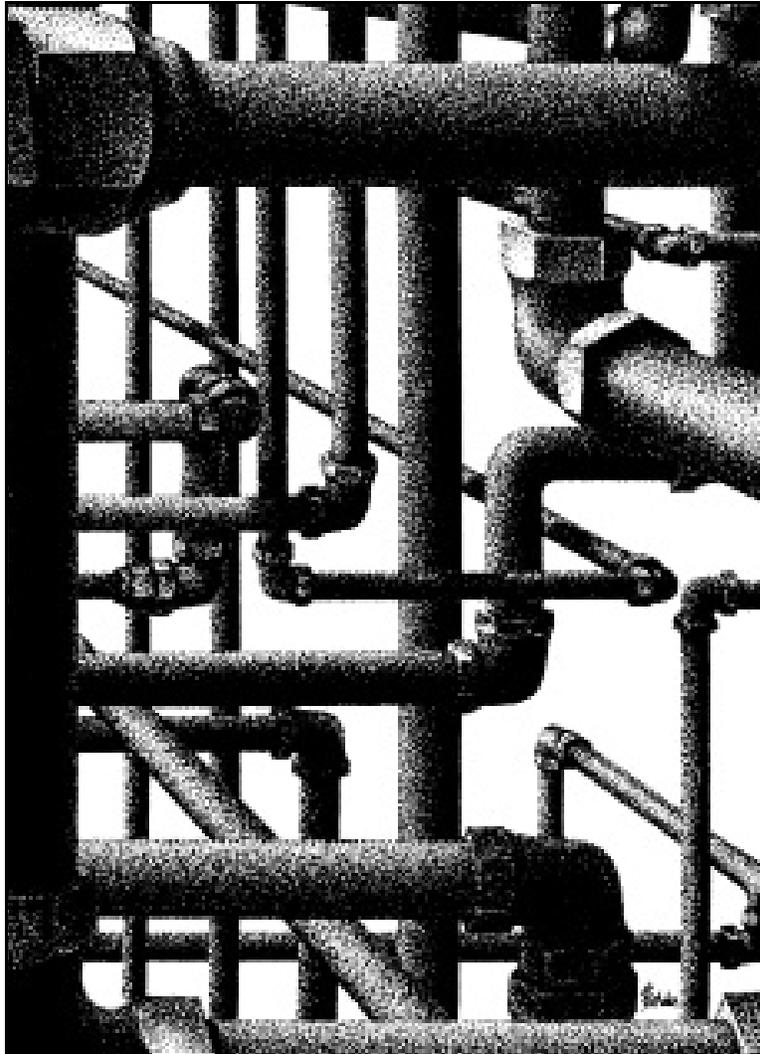
Thank you for your interest in our products, and for taking the time to view our demonstration software. We hope that you will be pleased with what you see during the demonstration. We are truly committed to providing products that ensure your success. If you have suggestions which might help improve GASWorkS to better meet your needs - *please,* let us know your ideas. Again, thank you for your interest...

Bradley B. Bean

GASWorkS™ 10.2   *Table Of Contents*

GASWorkS™ 10.2 — *Demonstration Guide*

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 that are 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 contained 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 individual calculation references.

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 at any time, we may be in.

By Telephone - (719) 578-9391

By Email - help@b3pe.com

By Website - www.b3pe.com

System/Hardware Requirements

GASWorkS will operate on any device configured to run Windows™ 7 thru Windows™ 11.

GASWorkS will operate on any device configured to support and execute the supported operating systems. Best performance will be achieved when 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 touch pad or similar pointing devices are 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.

Installing GASWorkS

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

Note...
> On Windows Vista and newer devices, you will need to be logged on as the Full Administrator or use the "*Run as administrator*" option when executing the installation file.

● Download the compressed folder (.zip) containing the setup executable file from our website, to your device. Navigate to the folder's location and open or explore it using Windows File Explorer.

● Double-click the setup file's name in the compressed folder. 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 the contents of each screen carefully and respond appropriately.

Note...
> On the License Agreement screen, read the terms of the License carefully and select the "*Agree to*" option. If you do not wish to accept the License Agreement, 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 whether it 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, begin its execution now.
>
> • The device must be connected to the internet the first time that the GASWorkS software is executed.
>
> • On Windows Vista and newer devices, 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 10 Demo* shortcut icon.

● The software will be started, and the GASWorkS Startup 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.
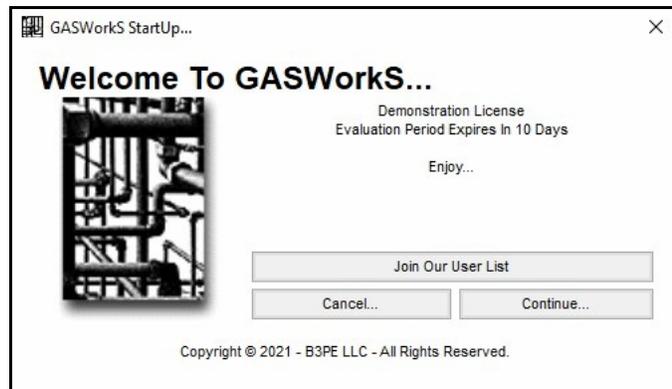
Example

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 provided in the following sections to create, edit, solve, and report your own 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 capacity of the system. Changes will be made to the system to allow the addition of the load. During the course of 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.

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 *Settings* item from the *Preferences* submenu of the *File* menu.
>
> • This example assumes that the settings, options, and default data values are set to the original 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 Model Selection screen will be displayed.

    ● In the *Model Name* data field, enter a name for the new model. For example, type **MyDemo** in the data field. When an appropriate *Model Name* has been entered, click the *Continue* command button to proceed.

The Graphic Data Interface (GDI) Window will be displayed. *Note* - If the GDI Window is not automatically displayed, select the *View/Edit* item from the *Graphics* menu to display the GDI Window.

Resize the GDI Window using the *Maximize GDI Window* icon from the *GDI Window Controls Toolbar*.
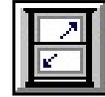
Note...

> • To assist in building a model using the drawing method, GASWorkS provides two "snap" methods which cause a pipe end to be snapped to an existing node. The first snap mode is referred to as "auto-snap". This is the method we have been using by holding down the *Shift* key while left-clicking the mouse when prompted for a node location.
>
> • The second method is referred to as 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 will be displayed around the mouse crosshairs. This circle represents the snap distance (a feature falling within the snap distance will be connected to).

Ensure that the Feature Snap is turned "on". Click on the *Feature Snap* icon from 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 model, the data values for the new features are set using 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 will need to be appropriately set when changing model types.

An alternative to using the Default Data Values to populate values for newly entered features, the data from the last similar feature in the model can be used to populate the values for a new feature. To use this method, select the *Use Last Record For Default Values* option in the Preference Settings.

Note...

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

To set the default data values...

● From the *Utilities* menu, select the *Set Defaults* item. The Default Data Values screen will be displayed.

● On the *Dimensional Units* data tab, set the following values:

Coordinates = **Feet** [Metres]
Customer Load = **Cfh** [M3h]
Diameter = **Inches** [Millimetre]
Efficiency = **Decimal**
Elevation = **Feet** [Metres]
Heating Value = **Btu/cf** [MJoules/m3]
Length = **Feet** [Metres]

Node Load = **Mcfh** [M3h]
Pipe Flow Rate = **Mcfh** [M3h]
Pressure = **Psi** [mBar]
Temperature = **Fahrenheit** [Celsius]
Velocity = **Feet/sec** [Metres/sec]
Viscosity = **Lbm/Ft-sec** [Centipoise]
Prompt Length = **Feet** [Metres]

● On the *Hydraulic Data* tab, set the following values:

Pipe Size/Type = **2P** [63P-11]
Pipe Equation = **IGT-Improved**
Pipe Efficiency = **0.95 Decimal**　　　　　　Allow Pipe Sizing = **Unselect**

Node Pressure = **0 Psi** [0 mBar]　　　　　Value Is Unknown = **Select**
Node Base Load = **-10 Mcfh** [-280 M3h]　　Value Is Unknown = **Unselect**
Node Elevation = **0 Feet** [0 Metres]
Node Temperature = **60 Fahrenheit** [15.6 Celsius]　Value Is Unknown = **Unselect**

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

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 = **Unselect**

● When all the values have been entered, click the *Close* command button to close the screen and save the settings.

Set The Preference Settings

● From the *File* menu, in the *Preferences* submenu, select the *Settings* item. The Preference Settings screen will be displayed.

> Use Last Record For Default Values = **Unselect**

● Click the *Close* command button to save the changes and close the screen.
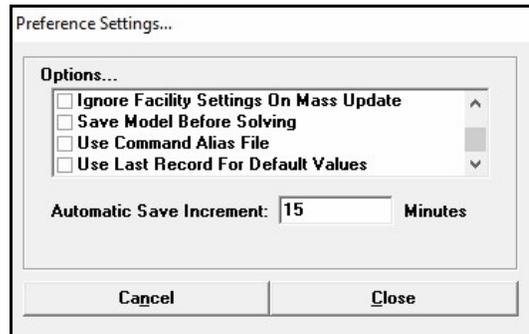
Set The Graphic Settings

When a new customer, node, or pipe feature is added to the model, the data for the new feature can automatically be displayed in the Data Panel after the addition. This option allows the User to make desired changes to the added feature's data during the entry command. The *Allow Data Entry During New Feature Entry* option, by default, will be turned "on" for easy editing while adding the new features.

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. A set of GDI Toolbar Icons are displayed along the left side of the GDI Window. When the mouse pointer is passed over a GDI Toolbar Icon, the toolbar will be expanded exposing the GDI Command Icons. The GDI Command Icons are used to execute many of the GDI Commands and are grouped by functionality.

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

The *GDI Command List* icon is located in the upper-right corner of the GDI Window. When selected, the GDI Command List will be displayed in the lower-right corner of the GDI Window. The GDI Command Line is located at the bottom of the GDI Window below where the mouse coordinate values are displayed.

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 GDI Window will display differently on different monitors. Let's resize it now so that it fits nicely on your monitor. Click the *Maximize GDI Window* icon from the *GDI Window Controls Toolbar* to fit the window to your monitor.
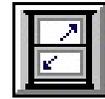
> *Note...*
>
> The GDI Window can also be manually sized by dragging the sides or corners of the window with the mouse. To save the current window size as the default for future models, select the *Settings* item from the *Graphics* menu. The Graphic Settings screen will be displayed. Click the *Save As Default* command button, click the *OK* command button to save changes, then click the *Close* command button to close the screen.

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 may be moved right or left and up or down by selecting one of the arrow symbols on the icon. The *Zoom To Fit*, *Zoom In*, and *Zoom Out* icons are located next to the *Scroll* icon. 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, and drawing the segment with the mouse. We will use each of these methods when constructing the example model. 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* icon from the *Graphic Construction Commands Toolbar*.

> ● 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 set to "No"

● The remaining values are acceptable. Click the *Apply Data Values* command button located at the bottom of the Data Panel to continue.

Note...

> The setting of the "*Known*" items indicate 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 demand nodes the pressure is generally 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 Node Data will be displayed in the Data Panel. The displayed values are acceptable. Either click the *Continue* command button, or right-click the mouse to continue.

● The Pipe Data 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 pipe, fitting, regulator, valve, compressor, and well data values. The User may add to or revise the data contained in any of the Property Tables.

GASWorkS™ 10.2 *Demonstration Guide*

Add The Second Pipe Using The Relative Distance & Angle Method

● Double right-click the mouse while the mouse pointer is in the GDI Display. The last GDI Command will be re-executed.

  ● For the *From Node Location*, right-click the mouse to accept the default From Node coordinates (**1500.000, 1000.000** [1150.000, 1000.000]). *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 Node Data will be displayed in the Data Panel. The displayed values are 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 Pipe Data 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 from the lower-left corner of the GDI Window to cause the GDI Image to be zoomed to fit into 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 and circle so that **Node 15** is within the circle, and 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 location for the node 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), and left-click the mouse to indicate the node's location.

● The Node Data will be displayed in the Data Panel. The displayed values are acceptable. Either click the *Continue* command button, or right-click the mouse to continue.

● The Pipe Data will be displayed in the Data Panel. The displayed values are 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.
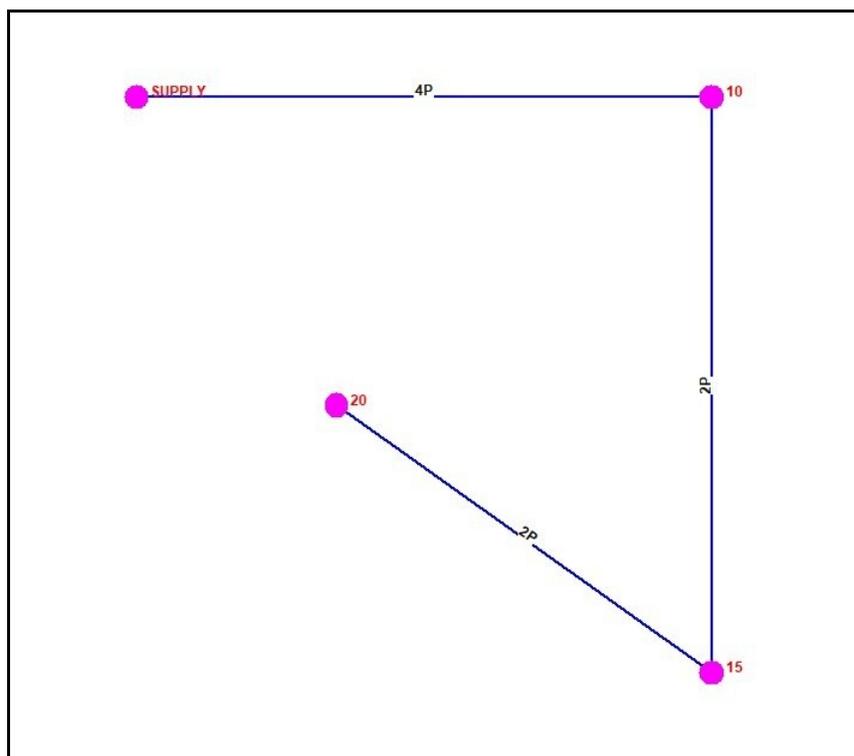

Figure 1

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*, move the mouse crosshairs and circle so that **Node SUPPLY** is within the circle, and 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 location for the node was not properly selected. To reselect the location, click the *Cancel* icon and reselect the *Add 2-Point Pipe* icon.

● The Pipe Data will be displayed in the Data Panel. The displayed values are acceptable. Either click the *Continue* command button, or right-click the mouse to continue.

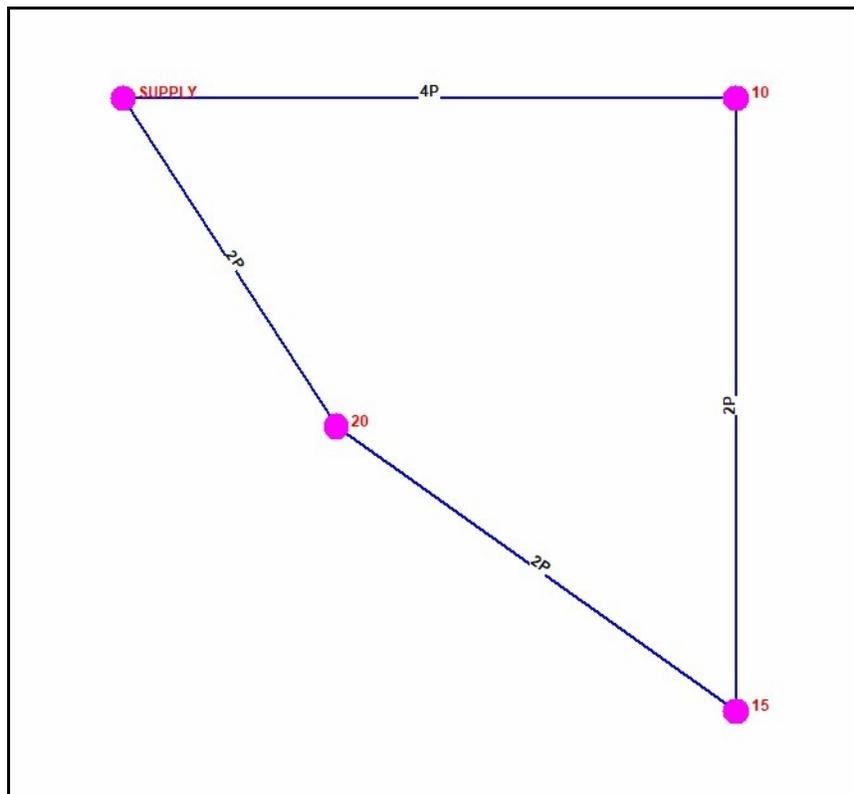After entering the last pipe, the model should look similar to Figure 2.

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...
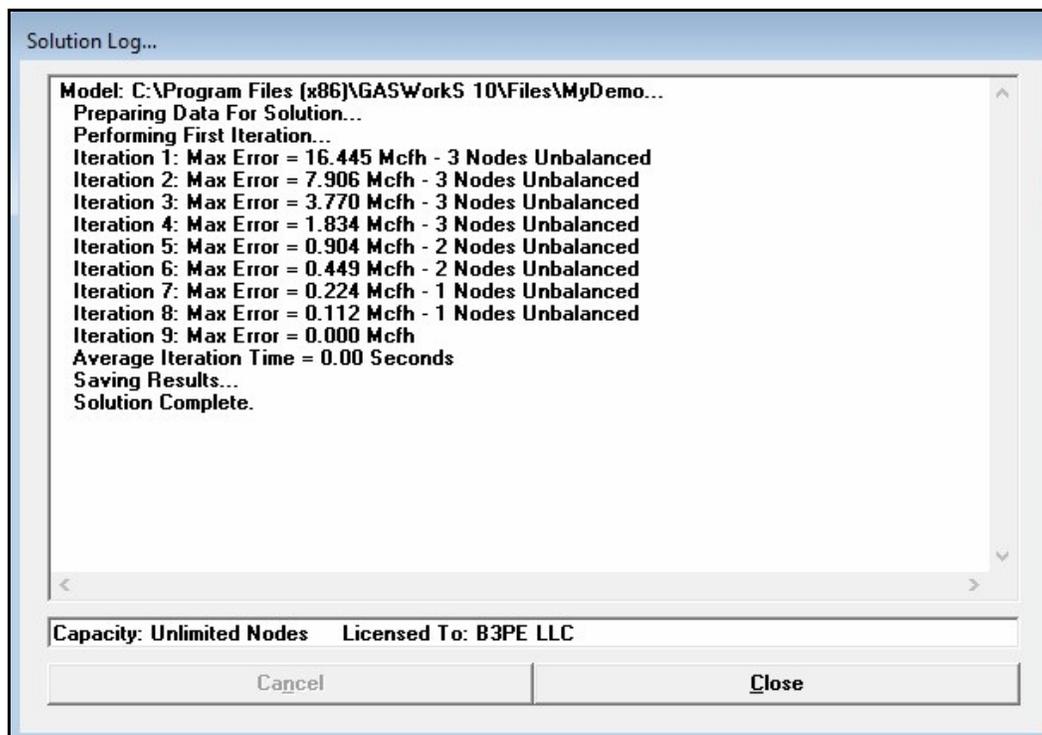
● Click the *Solve* icon from the lower-right corner of the GDI Window. The Solution Data screen will be displayed. On the *Other Settings* data tab, set the following values:

Base Pressure = **14.73 Psi** [1016 mBar]
Base Temperature = **60 Fahrenheit** [15.6 Celsius]

● For this example, the remaining default solution parameters are 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.

```
Solution Log...

Model: C:\Program Files (x86)\GASWorkS 10\Files\MyDemo...
Preparing Data For Solution...
Performing First Iteration...
Iteration 1: Max Error = 16.445 Mcfh - 3 Nodes Unbalanced
Iteration 2: Max Error = 7.906 Mcfh - 3 Nodes Unbalanced
Iteration 3: Max Error = 3.770 Mcfh - 3 Nodes Unbalanced
Iteration 4: Max Error = 1.834 Mcfh - 3 Nodes Unbalanced
Iteration 5: Max Error = 0.904 Mcfh - 2 Nodes Unbalanced
Iteration 6: Max Error = 0.449 Mcfh - 2 Nodes Unbalanced
Iteration 7: Max Error = 0.224 Mcfh - 1 Nodes Unbalanced
Iteration 8: Max Error = 0.112 Mcfh - 1 Nodes Unbalanced
Iteration 9: Max Error = 0.000 Mcfh
Average Iteration Time = 0.00 Seconds
Saving Results...
Solution Complete.

Capacity: Unlimited Nodes    Licensed To: B3PE LLC

         Cancel                              Close
```

Note...

> The Solution Log displays the progress of the Solution routine. During a normal solution the node errors should start relatively large and get smaller as the Solution progresses, until the node error is less than the specified convergence tolerance. The last line of the Solution Log should indicate "Solution Complete". If the Solution is not complete, recheck the model creation steps, correct any errors you encounter, then re-solve the model.

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

Notice that the *Solve* icon has changed. The scale is now blue and balanced indicating that the model is balanced.
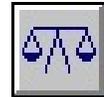
View The Analysis Results

Now let's review the analysis results. In addition to its various Report routines, GASWorkS also provides several graphical methods for viewing both the model data and the analysis results. We will look at the Report routines later, for now let's use some of the 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* icon from the *Display Controls Toolbar*.

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

● Click the *Text Display Settings* icon from the *Display Controls Toolbar*. The Text Display Settings screen will be displayed.

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

Display Node Text Items = **Select**
Transparent Font = **Unselect**

Node Pressure = **Select**

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

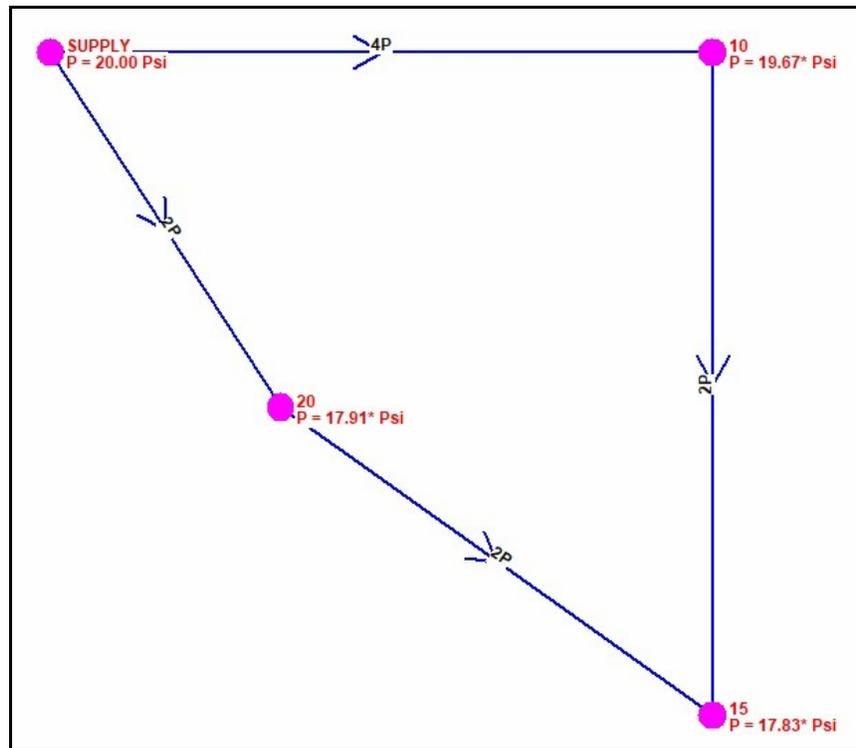The pressure values and flow directions should be similar to those shown in Figure 3.

Figure 3

Note...

• The displayed pressure values represent the estimated pressure for each node point. As a result of "drawing" in the third and fourth pipe segments, each example model is created slightly different. 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 that an error has occurred during the creation of the model. If this is the case, you might want to retrace the creation steps to identify the problem.

• The analysis results for an individual pipe or node can be viewed and printed using the *Edit Node Data* and *Edit Pipe Data* commands, or by simply clicking the mouse on the desired feature.

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 Pipe* icon from the *Graphic Edit Commands Toolbar*.

● For the *Select Pipe To Tap*, 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 Node Data 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 are 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.
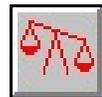
Re-solve the model by clicking the *Quick Solve* icon from the lower-right corner of the GDI Window. The pressure value for Node NEWTAP will be calculated and displayed on the model.

In Figure 4, notice that some of the pressure values are negative (less than zero). This indicates that the system has "run-out-of-gas". In other words, the increased demand caused by the new load, is greater then the system's capacity.
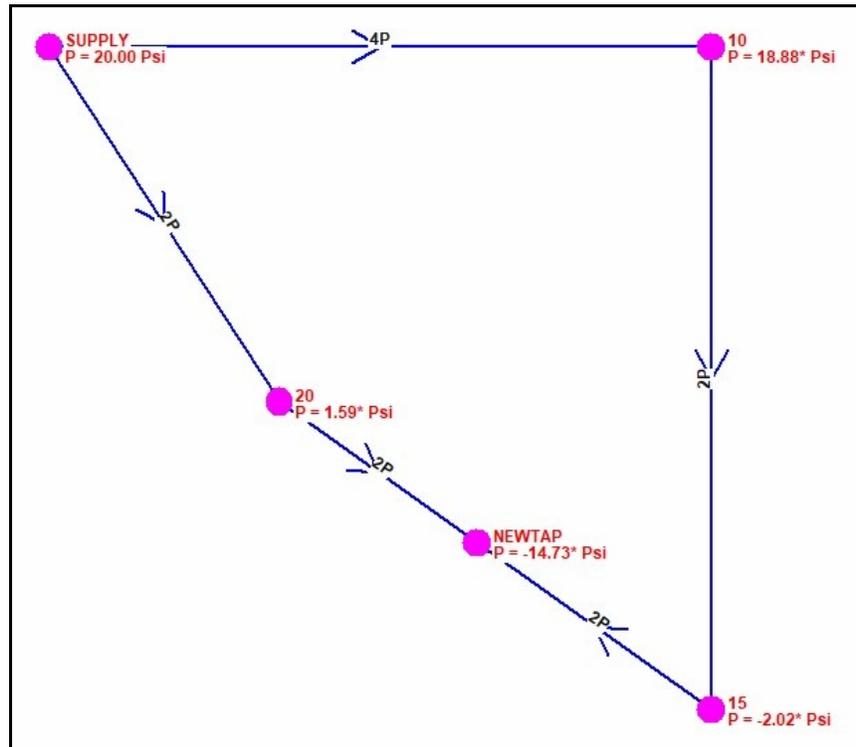

Figure 4

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 try to find the pipes with the highest pressure drop.

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, all of the nodes where the pressure value was less than 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 the pipes based on the their pressure drop values.

● Click the *Color Display Settings* icon from the *Display Controls Toolbar*. The Color Display Settings screen will be displayed.
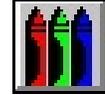
● On the *Range Colors* data tab, set the following values:

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 as seen in Figure 5.
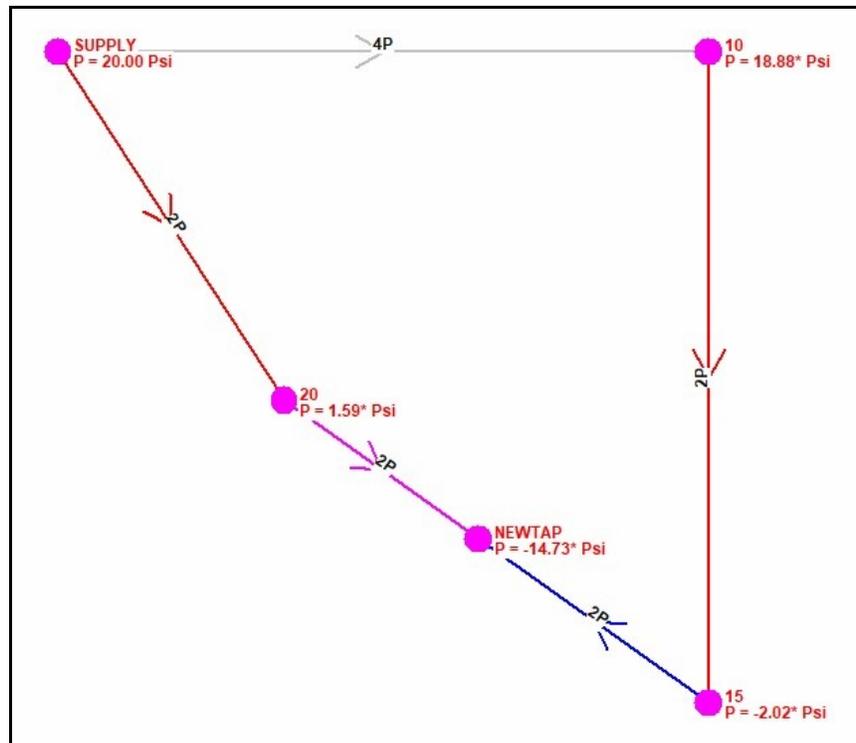

Figure 5

As a reminder of the range color values, click the *Display Color Legend* icon from the *Display Controls Toolbar*. The Color Legend will be displayed using the calculated color and value ranges.

You may move the Color Legend to a convenient location. The Color Legend can be resized by dragging its sides or corners with the mouse. To turn "off" the display of the Color Legend, click the *Display Color Legend* icon from the *Display Controls Toolbar* again.

Note...

> In this case, the pipes with the highest pressure drop are colored "red". The segments between Node 10 and Node 15 and between Node SUPPLY and Node 20 might both be colored "red" or one or the other might only be colored "red". 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 Data

During some of the 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 increase 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 to the pipe segment and left-click the mouse.

    - The Pipe Data 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 from the lower-right corner fo the GDI Window. The Solution Data screen will be displayed.
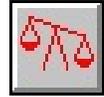
    ● On the *Solution Data* tab, set the following values:

Update Color Coding = **Select**

● 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 that we made fixed the problem, or 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) back 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 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.

Record Number	Name	Internal ID Number	Pressure	Pressure Units	Pressure Known	Base Load	External Load	External Load (Fixed)	Total Load
1	SUPPLY	2	20.00	Psi	Yes	70.010	0.000	0.000	70.010
2	10	4	19.32	Psi	No	-10.000	0.000	0.000	-10.000
3	15	6	11.49	Psi	No	-10.000	0.000	0.000	-10.000
4	20	8	19.07	Psi	No	-10.000	0.000	0.000	-10.000
5	NEWTAP	10	10.59	Psi	No	-40.000	0.000	0.000	-40.000

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** [2000]

● 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 the lowest to highest value.

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 there were only five nodes in the model, so all records were included in the selection set, ranking them from the lowest to the highest value. 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 Flow Rate, Pressure Drop, Velocity, and Volume 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 contents of the report by expanding and collapsing columns, and by creating query based selection sets. There are also several routines provided for sorting and browsing through the report contents.
>
> • Because of the amount of data contained in the reports, the report width will generally be greater than a printed page width. When using the Print routine, a message will be displayed stating that the report width is greater than the page width. An option will be given to cancel the Print routine or continue. Generally, you will want to select the continue option. If the Print routine is continued, the report will be printed on multiple pages.
>
> • 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 Columns screen will be displayed. The selected column names represent the columns that are currently displayed. Unselect some of the columns. Use a proportional font like "Arial Narrow". Use a font size of 9 points or smaller. And set the printer orientation to "Landscape".

What To Do Next

These last few steps demonstrate the most powerful use of GASWorkS - the ability to study the affect of system modifications before they are implemented. By analysis of these effects using the model, changes which 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 installation of incorrectly sized facilities in the field.

There are several prebuilt example models distributed with the software. Refer to the Help Guide located in the *Help* menu for descriptions and instructions for viewing the various example models. These prebuilt examples include applications using regulators, hydraulic valves, fittings, compressors, and wells, and demonstrate many of the features associated with the software.

This simple example provided a brief overview of the basic routines found in the GASWorkS software. There are many routines that were not used in the example - the Import and Export routines, attribute valves, tracing, data checking, use of the Pipe Sizing routine, and use of the customer data features were all absent from the demonstration example. Perhaps you may wish to explore some of these features on your own. Information about these features can be found in the User's Manual or the Help Guide. These documents may be downloaded from the *Documents* section of our website, www.b3pe.com.

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 software you wish to remove it, select the *Uninstall GASWorkS 10 Demo* item from the *GASWorkS* Programs List.

● 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 10 Demo\ini**
  • 64-bit machines - **C:\Program Files (x86)\GASWorkS 10 Demo\ini**