# GASWorkS™ 10.0 Example   *Town DXF*

Summary

In this example, a model will be created from an existing DXF file. The piping drawn in the DXF file will be imported to create the model schematic. The DXF file will also be used as a background image to the model schematic. The Trace routine will be used to perform a data check of the imported model. The model consists of a high pressure supply line, a regulator at the town border station, and a medium pressure distribution system. The resulting model will be solved and the results reviewed.
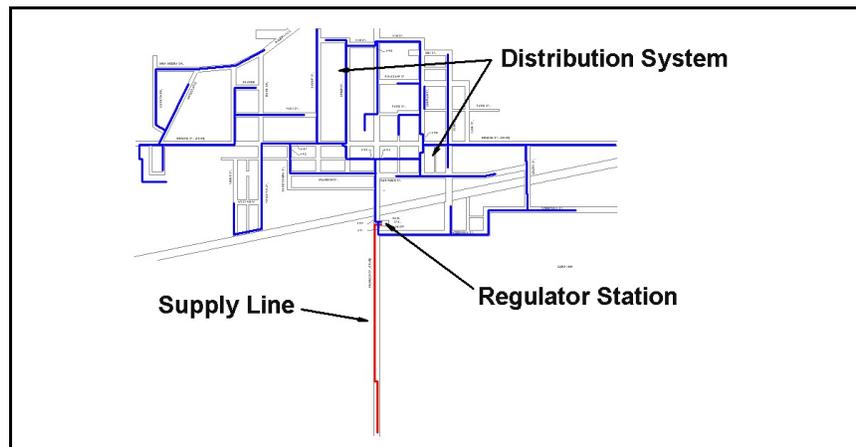
Steps

The following assumes that GASWorkS has already been started. If a model is already open, close it by selecting the *Close* menu item from the *File* menu list. Use the following procedure to work this example...

1) Import The DXF File

● From the *Utilities* menu list, select the *Import* menu item. The File Selection screen will be displayed. Enter a *Filename* for the new model, then select the *Continue* command button.

> ● The Import screen will be displayed. Select the *Set Default Data Values* command button. The Default Data Values screen will be displayed.

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

>>> Coordinates = **Feet**
>>> Customer Load = **Cfh**
>>> Length = **Feet**

Node Load = **Mcfh**
Pipe Flow Rate = **Mcfh**

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

Pipe Size/Type = **99**   (*Note* - This value is used to help identify pipes where the
diameter is not automatically assigned during the import.)
Pipe Equation = **IGT-Improved**
Pipe Efficiency = **0.95 Decimal**
Allow Pipe Sizing = **Unselect** (Uncheck)

Node Pressure = **0 Psi**	Value Is Unknown = **Select** (Check)
Node Base Load = **-0.5 Mcfh**	Value Is Unknown = **Unselect** (Uncheck)

(*Note* - The load value is arbitrary - just used to get started)
Node Elevation = **300 Feet**
Node Temperature = **60 Fahrenheit**   Value Is Unknown = **Unselect** (Uncheck)

● Select the *Close* command button to close the screen and save the changes.

● On the Import screen , from the *File Type* list, select **DXF (Drawing Exchange Format) File**, then select the *Import* command button.

● The File Selection screen will be displayed. Use the *Drives* and *Directories* lists to navigate to the directory containing the desired file - **town.dxf**. Select the file, then select the *Continue* command button.

● The DXF Import Specifications screen will be displayed.

● On the *Pipe Settings* data tab, select the *Clear Pipe Layer Assignments* command button. At the "Really Clear..." message, select the *Yes* command button. Set the following values:

Pipe Layers -
Layer Assignment = Pipe Size
Main_2P = **2P**
Main_2S = **2S**
Main_3P = **3P**
Main_4P = **4P**
Main_6P = **6P**

Fuzzy Tolerance = **1**
Arc Resolution = **0**
Pipe Length Units = **Feet**

Attribute Valve Layer = **Empty (no selected value)**

● On the *Other Settings* data tab, set the following values:

Drawing Coordinate Units = **Feet**
Origin Shift: X = **0**   Y = **0**
Model Coordinate Units = **Feet**

Customer Layer = **Empty (no selected value)**

● Select the *Continue* command button to import the data.

● When the process is complete, on the Import screen select the *Close* command button.

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

● Resize the GDI Window using the *Maximize GDI Window* icon from the *GDI Window Controls Toolbar*.
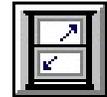
2) Set The Graphic Settings

Let's set some graphic display values and options.

● From the *Graphics* menu list, select the *Settings* menu item. The Graphic Settings screen will be displayed.

● In the *Settings* section, set the following values:

Node Symbol Display Limit = **10000**
Node Symbol Size = **1% Of Display Width**

● In the *Options* section, set the following items:

Query During New Feature Entry = **Unselect** (Uncheck)

● Select the *Close* command button to close the screen and save the changes.

3) Attach The Background Image

Let's attach a background image to display the road right-of-ways and other base data.

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

    ● Select the *Attach New Image* command button. The Attachment File Type screen will be displayed. Select **DXF - Drawing Exchange Format Style CAD File**, then select the *Continue* command button.

    ● The File Selection screen will be displayed, listing the available DXF files. Use the *Drives* and *Directories* lists to navigate to the directory containing the desired file - **town.dxf**. Select the file, then select the *Continue* command button.

● On the Background Image Settings screen, select the *Apply* command button to attach the DXF image to the model.

Note - If the background image is not displayed in the GDI Display, select the *Display Background Image* icon from the *Display Controls Toolbar*. The background image should now be displayed.
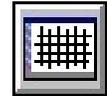
4) Check The Model Connectivity

Let's use the Trace routine to check the model connectivity.

● Select the *Trace* icon from the *Utility Commands Toolbar*.

    ● The Trace Specifications screen will be displayed. Set the following values:

        Trace Style = **Trace All Directions**
        Start Trace At = **A Node**
        Trace Highlight Color = **Red**   (Click in the color box. The Color Palette will be displayed. Select the color Red, then select the *OK* command button.)

        Reset Feature Colors Before Tracing = **Select** (Check)

    ● Select the *Trace* command button.

● For the *Starting Node*, select the node on the very south end of the pipe extending south at the center of the system (this is the supply end of the supply line), **Node 190**. If everything is connected, the trace should color the entire system red. Are all of the pipes colored red? No.

5) Edit The Model Connectivity

A discontinuity should appear at the north end of the supply line. This is the location of the town border station and regulator.

● Zoom into the area where the color changes using the *Zoom Window* icon from the *View Controls Toolbar*.
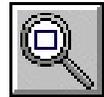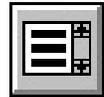
Note - Before adding the regulator, let's turn "on" the Node Snap Mode option. Select the *Node Snap* icon from the lower-left corner of the GDI Window until the background of the icon image is darkened.

We now need to add a regulator between the supply line and the distribution system.

● Select the *Add Regulator* item from the GDI Command List.

   ● For the *From (Upstream) Node Location*, select the end of the supply line (the red line - **Node 185**) by placing the snap "target" over the node and left-clicking the mouse.

   ● For the *To (Downstream) Node Location*, select the end of the distribution system main line (**Node 175**) by placing the snap "target" over the node and left-clicking the mouse.

   ● For the *Regulator Size & Type*, select **Universal** and press the *Enter* key.

   ● For the *Set Pressure Units*, select **Psi** and press the *Enter* key.

   ● For the *Set Pressure*, type **20** and press the *Enter* key.

   ● For the *Facility Type*, select **UnSpecified** and press the *Enter* key.

Note - If the new pipe is not displayed with a regulator symbol, turn "on" the piping symbols by selecting the *Display Piping Symbols* icon from the *Display Controls Toolbar*. The new pipe should now be drawn as a regulator symbol. The inlet side of the regulator is always to the left of the control head as it is pointing up. The inlet side of the regulator must be the same as the From Node - the inlet must always be connected to the high pressure (upstream) system.

Bradley B Bean PE   *ENGINEERING & SOFTWARE*

6) Recheck The Connectivity

Try the Trace routine again to see if the connectivity is now continuous.

● Select the *Trace* icon from the *Utility Commands Toolbar*. Follow the previously described steps to trace from the upstream side (south end) of the regulator.
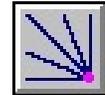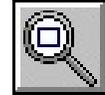
Zoom the entire GDI Image to fit in the GDI Display.

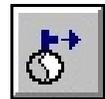● Select the *Zoom To Fit* icon from the lower-left corner of the GDI Window.
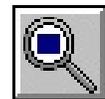
Are there any other discontinuities? There should be one more. The first lateral pipe going north from the 3 inch main east of the regulator station. Use the Zoom Window icon from the *View Controls Toolbar* to view the area around the south end of the lateral. The lateral (**Node 5**) is not connected to the horizontal header pipe (Pipe 35 to 170).

Let's fix it using the *Tap Pipe* command.

● Select the *Tap Pipe* icon from the *Graphic Edit Commands Toolbar*.

● For the *Pipe To Tap*, left-click the mouse on **Pipe 35** to **170**.

● For the *Tap Location*, left-click the mouse on **Node 5**.

● **Pipe 35** to **170** will now be split into two pipe segments with the segments being snapped to the existing node (**Node 5**).

● Retrace from **Node 5**. All pipes connected to the node should now be colored in the *Trace Highlight Color* (Red).

Let's check the connectivity one more time.

● Select the *Zoom To Fit* icon from the lower-left corner of the GDI Window. Are there any other discontinuities? There shouldn't be. If there are still discontinuities an error has probably occurred during the previous steps. Review and re-work them to correct the issue.

7) Set The Non-Default Pipe Data

Set the pressure on the supply end of the supply line.

● Left-click on **Node 190** (on the south end of the supply line). *Note* - If you cannot select the node, use the *Edit Node Data* command.

> ● The Node Data will be displayed in the Data Panel. In the *Hydraulic Data Items* section, set the following values:

> > Pressure = **150**
> > Pressure Units = **Psi**
> > Pressure Known = Yes   (*Note* - The *Base Load Known* value should automatically be set to **No**.)

> ● Select the *Apply Data Values* command button to save the changes.

Before continuing, let's reset the feature colors. Type **RESET** on the GDI Command Line and press the *Enter* key.

8) Solve The Model

The model should now be ready to solve. Select the *Solve* icon from the lower-right corner of the GDI Window. The Solution Data screen will be displayed. Review the solution parameters, then select the *Solve* command button. The Solution Log will be displayed. Review the results, then select the *Close* command button to close the log.
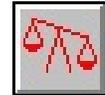
9) Review The Solution Results

Review the solution results by displaying the flow arrows and pressure values.

● If the flow arrows aren't displayed, select the *Display Flow Arrows* icon from the *Display Controls Toolbar*.

● Display the pressure values by selecting the *Text Display Settings* icon from the *Display Controls Toolbar*. On the Text Display Settings screen, on the *Node Items* data tab, select (check) the *Display Node Text Items* option, and select (check) the *Pressure* item. Select the *Apply* command button to update the display. Are the pressures OK?

● You can also color-code the results using the Color By Range routine in the Color Display Settings screen. A good way to use this command is to color-code the pipes based on a minimum pressure value.
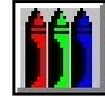
Note - Once the model successfully solves you would usually proceed with the model refinement and calibration. However for this example, this is the end of the instructions. You can view the help topics for instructions for performing these other tasks.

Notes & Considerations

● In this example, an existing DXF file was used as the basis for the model. When the file was imported, nodes were automatically assigned at the beginning and end of each line, polyline, or arc segment found on the specified pipe layer. These segments were turned into pipes. The pipe sizes were assigned based on the layer assignments made in the DXF Import Specifications. Pipe lengths were set to the scaled lengths based on the coordinate values found in the drawing.

● The DXF file used for this example was fairly clean. The pipes were snapped together and mostly broken at each intersection. The pipes were accurately grouped on the appropriate layers. Drawings are not always this well prepared. GASWorkS provides various tools for editing the piping data after it has been imported. However, it is best to make any graphic revisions in the original DXF file. That way if the drawing ever needs to be imported again, the corrections will not need to be repeated.

● In this case we assumed that the drawing was to scale, however we didn't check it. If the drawing wasn't to scale we could scale and rotate it after it is imported. Or ideally, we would go back to the original drawing and rescale it in the CAD system used to create the DXF. This would ensure that next time we used the file it would be to scale. If we scale the drawing after it has been imported, the piping network will not properly overlay on the background image unless it is also scaled.

● Only a cursory check of the pipe sizes and connectivity was made in this example. Before actually using a model created by importing, each pipe size and the pipe connectivity should be carefully checked in detail.

● Only generic loads were applied when the model was imported. The actual system loads would need to be determined and accurately assigned before using an imported model to make any decisions.

● As with any existing system, once the model data has been checked and the loads accurately assigned, the model should be calibrated against actual field data. The calibrated model can then be used as the basis for additional analysis and what-if scenarios.

● Here are a few tips to consider when preparing a drawing for use with GASWorkS:

　　• Create the drawing to scale.
　　• Use decimal notation for lengths – avoid using architectural notation.

      • Snap all line (pipe) ends together.
      • Break lines (pipes) at intersection were the intersecting lines (pipes) are connected.
      • Only place lines that represent pipes on the pipe layer – avoid including symbols or blocks on the pipe layer.
      • Where practical, group the piping on separate layers based on size and material type.
      • If attribute valves are included in the DXF, place them on a separate layer using a consistent block symbol to identify their locations.

● In this example we installed one regulator at the end of the supply line. This regulator generically represented the "regulator station". For most applications of this nature, this representation is adequate. If a more detailed analysis of the station is required, a more detailed model of the station could be created separately or additional detail could be added to the system model.

● When including a regulator in a model, the set pressure must be entered. The outlet and inlet pressures can be "Known" or "Unknown". However, if the outlet pressure is "Unknown", its calculated value will be based on the set pressure and the regulator characteristics. Generally, the load value for the inlet and outlet of a regulator is set to zero (0) and "Known".

Bradley B Bean PE   *ENGINEERING & SOFTWARE*
9