

## Summary

In this example, a new model will be created for a small subdivision. A DXF file of the subdivision plat will be used as a background image to the model schematic. The model will be created by tracing over the image of the plat. Customers will be added along the new mains. The resulting model will be solved and the solution results reviewed.

## Steps

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

### 1) Create The New Model

- From the *File* menu, select the *New* item. The Model Selection screen will be displayed. Enter a *Model Name* for the new model, then click the *Continue* command button.

- 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*.



## 2) Set The Preference Settings

---

For this model, when we are entering new features, it would be good for the data values for each new item to be same as the values used for the last entered similar feature. To make this happen we need to select a specific option in the Preference Settings.

- From the *File* menu, in the *Preferences* submenu, select the *Settings* item. The Preference Settings screen will be displayed. Set the following values:

Use Last Record For Default Values = **Select**

- Click the *Close* command button to save the change.

## 3) Set The Graphic Settings

---

Before building the model, let's check that a few specific graphic setting values and options are set appropriately.

- From the *Graphics* menu, select the *Settings* item. The Graphic Settings screen will be displayed.
  - In the *Settings* section, set the following values:

Customer Symbol Display Limit = **10000**  
Customer Symbol Size = **30** (Coordinate Units)  
Node Symbol Display Limit = **10000**  
Node Symbol Size = **1 % Of Display Width**  
Refresh Increment = **10 % Of Image**

- In the *Options* section, set the following values:

Allow Data Entry During New Feature Entry = **Unselect**  
Automatically Pan & Zoom = **Select**  
Automatically Repaint GDI Image = **Select**  
Display Vertex Symbols = **Select**  
Ignore Pipes Outside GDI Display = **Unselect**

- Click the *Apply* command button to close the screen and save any changes.

## 4) Set The Default Data Values & Units

---

Although we have selected the *Use Last Record For Default Values* preference setting option to handle the data values for new features, we should also set the “global” default data values and dimensional units.

- 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**  
Customer Load = **Cfh**  
Node Load = **Mcfh**  
Pipe Flow Rate = **Mcfh**

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

Pipe Size/Type = **1-1/4P Inches**  
Pipe Equation = **IGT-Improved**  
Pipe Efficiency = **0.95 Decimal**      Allow Pipe Sizing = **Unselect**

Node Base Load = **0 Mcfh**      Value Is Unknown = **Unselect**

*Note* - Because we will use “customers” to apply the load to this model

Node Base Elevation = **0 Feet**

- On the *Gas Properties* data tab, check the values for appropriateness.

- Click the *Apply All Values* command button to apply the new values to the model. A message will be displayed after the values have been applied. Click the *OK* command button to clear the message.

*Note...*

Use caution when using the “*Apply*” command buttons on the Default Data Values screen. Applying values to an existing model will change all pipe and node values which have an associated default data value. Data that has already been set in the model, will be overwritten with the default values.

- Click the *Close* command button to close the default data screen and save the changes.

## 5) Attach The Background Image

For this example we will attach a background image to model, then trace the new piping system using the background image as a “template”.

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

- Click the *Attach New Image* command button. The Attachment File Type screen will be displayed. Select **DXF - Drawing Exchange Format Style CAD File** from the list, then click the *Continue* command button. The File Selection screen will be displayed. Use the *Drives* and *Directories* lists to navigate to the directory containing the desired file - **plat.dxf**. Select the file, then click the *Continue* command button.

- On the Background Image Settings screen, set the following values:

Scale Factor = **1**  
Origin Shift: X = **0** Y = **0**

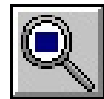
*Note* - The layers section contains the layers found in the DXF file. Individual layers can be turned “on” or “off” using this list. For now, leave all of the layers at their current settings.

Line Weight = **1**  
Convert Colors To Gray = **Select**

- Click the *Apply* command button to close the screen and attach the DXF image to the model.

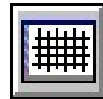
## 6) Zoom To Display The Background Image

- Zoom the GDI Image to fill the display using the *Zoom To Fit* icon from the lower-left corner of the GDI Window.



*Note...*

If the background image is not displayed in the GDI Display, click the *Display Background Image* icon from the *Display Controls Toolbar*. The background image should now be displayed.



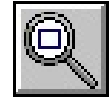
## 7) Check The Scale Of The Background Image

---

It is always good practice to check the scale of the background image before tracing over it.

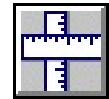
Zoom into a portion of the GDI Image where the right-of-way (ROW) can be measured.

- Click the *Zoom Window* icon from the *View Controls Toolbar*. Select a zoom window around “Golden Eagle Drive” so that the distance between “Lot 76” and “Lot 82” may be measured.



Measure the distance between the ROW lines.

- Click the *Measure Distance* icon from the *Utility Commands Toolbar*.
  - For the *First Point*, left-click on one of the ROW lines.
  - For the *Next Point*, left-click on the opposite ROW line approximately perpendicular to the first point. Right-click the mouse to end the point selection.
  - The Measured Values screen will be displayed. The Absolute Distance should be about 50 Feet. Is the distance correct? Click the *OK* command button to clear the message.



## 8) Turn “Off” Some DXF Layers

---

You might have noticed that the background image has a lot of text displayed in it. Let’s turn “off” some of the unnecessary information (such as the text layer).

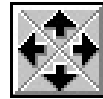
- From the *Graphics* menu, select the *Background Settings* item. The Background Image Settings screen will be displayed.
  - In the *Attached Background Image Files* list, ensure that the **plat.dxf** file is highlighted. The lower half of the screen will display the layers and settings for the selected (highlighted) file.
  - In the layers section, unselect the **PP-TEXT** item.
- Click the *Apply* command button to apply the layer changes to the background image.

## 9) Enter The Piping System

It is now time to “draw” the piping system. The final model should look similar to the figure on the first page.

*Note...*

You will probably need to use the various *Zoom* and *Pan* commands and methods to adjust the display of the background image as you are adding the new pipe features. Only certain methods can be used while a “drawing” command is in use. The mouse wheel can be used to zoom in and out. The *Scroll* icon can be used to pan the image. Most of the other pan and zoom methods should be avoided during a drawing command.



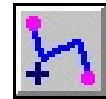
*Note...*

To help ensure that the new pipes are all connected at their ends, let’s turn “on” the Feature Snap. Click the *Feature Snap* icon from the lower-left corner of the GDI Window until the icon background is darkened. When the Feature Snap is “on”, a “target” will be displayed around the mouse cursor when a feature location is requested. If an existing feature is found within the target area, the new feature will be “snapped” to the existing feature.



For this model, because of the curved streets, it would probably be best to use a polyline type pipe. To add a pipe using this pipe style...

- Click the *Add Polyline Pipe* icon from the *Graphic Construction Commands Toolbar*.
  - For the *From Node Location*, select the desired location by left-clicking the mouse.
  - For the following vertices, continue to select the desired locations by left-clicking the mouse.
  - After the vertex representing the To Node has been entered, right-click the mouse to end the command.



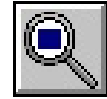
Repeat the previous steps to add the remaining pipe features.

*Note...*

To draw the pipe in the cul-de-sac, you can either use a polyline or arc style pipe. To use an arc style pipe, end the polyline pipe at the entrance to the cul-de-sac. Use the *Add Arc Pipe* command. Select the last node on the polyline pipe as the From Node and select the To Node Location, then drag the curve to fit the cul-de-sac.



- After all of the pipe features have been entered, zoom the entire GDI Image by clicking the *Zoom To Fit* icon from the lower-left corner of the GDI Window. The piping configuration should look similar to the figure on the first page.

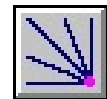


### 10) Trace The Piping System

---

It is good practice to check the connectivity of the newly entered pipes before proceeding further with the model building. One method to do that, is to use the *Trace* command.

- Click 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** Note - Click in the color box. The Color Palette will be displayed. Select the color “Red”, then click the *OK* command button.

- Click the *Trace* command button.
- For the *Starting Node*, select any node in the system. If everything is connected, the trace should color the entire piping system “Red”. Are all of the pipes colored red? No.

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

- Click the *Tap Pipe* icon from the *Graphic Edit Commands Toolbar*.



- For the *Pipe To Tap*, left-click the mouse on **Pipe 5 to 10**.
- For the *Tap Location*, left-click the mouse on **Node 15**.

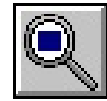
Pipe 5 to 10 will now be split into two pipe segments with the segments being snapped to the existing node (Node 15).

Let’s check the connectivity one more time.

- From the GDI Command List, select the *Quick Trace* item. All pipes should now be colored in the *Trace Highlight Color* (Red).



- Click 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.



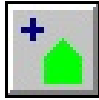
Before continuing, let's reset the feature colors. Type **RESET** on the GDI Command Line and press the *Enter* key.

### 11) Enter The Customer Locations

---

After the piping has been entered and checked, it is time to add the customers to the model. We will do that by adding one customer, setting its load value, then adding the remaining customers. The customer locations should look similar to Figure 1.

Add the first customer.

- Click the *Add Customer* icon from the *Customer Commands Toolbar*.A square icon with a blue plus sign and a green house shape.
- For the *Customer Location*, left-click at the desired location.
- For the *Supply Main*, left-click on the desired main (pipe).
- Left-click on the customer symbol to display the Customer Data in the Data Panel. In the *Hydraulic Data Items* section, set the following values:


Per Unit Load = **-100** *Note* - Negative to indicate gas leaving the system.

Load Units = **Cfh**

*Note* - This should already be set based on the previously set Default Data Values.

- Click the *Apply Data Values* command button to save the changes.

Continue adding the rest of the customer features.

- Click the *Add Multiple Customers - Auto Assign Main* icon from the *Customer Commands Toolbar*.A square icon with a blue plus sign and a network of colored lines (purple, green, blue) representing a customer assignment.
- Select the location for each customer feature (see the figure on the first page). The customer will then be assigned to the main nearest its selected location. When the last customer has been added, right-click the mouse to end the command.



*Note...*

If some customer to main assignments need to be revised, use the *Reassign Customer Supply Main* command. Select the customer to be revised, then select the new supply main.

## 12) Spot Check The Data

We can check the node loads by displaying their value on the GDI Image.

- 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 values:



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

Node Load, Total Adjusted = **Select**

- Click the *Apply* command button to update the GDI Image.

There should be a non-zero load value for every node but the supply node (most northern node in the figure on the first page).

## 13) Enter The Supply Pressure

Before the model can be solved, at least one pressure value must be known. In a model like this one, the pressure at the node representing the supply point is usually set as “known”. To set the supply pressure for the supply node...

- Left-click the mouse on the supply node (most northern node in the figure on the first page). The Node Data will be displayed in the Data Panel. In the *Hydraulic Data Items* section, set the following values:

Pressure = **14.5**

Pressure Units = **Psi**

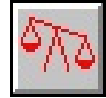
Pressure Known = **Yes** *Note - The Base Load Known will automatically be set to “No”.*

- Click the *Apply Data Values* command button to save the changes.

## 14) Solve The Model

---

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



## 15) Review The Results

---

- Review the solution results by displaying the flow arrows and pressure values.
- If the flow arrows aren't displayed, click the *Display Flow Arrows* icon from the *Display Controls Toolbar*.
- Display the pressure values by clicking the *Text Display Settings* icon from the *Display Controls Toolbar*. On the Text Display Settings screen, on the *Node Items* data tab, select the *Node Pressure* item. Click the *Apply* command button to update the display. Are the pressures OK?



**Note...**

The flow arrows might be obscured by the pipe size text. If so, turn the pipe text display “off” by unselecting the *Display Pipe Text Items* option on the *Pipe Items* data tab on the Text Display Settings.

All of the flow arrows should be pointing away from the supply node. Are they? If not, then the load values were probably entered as a positive number instead of a negative (-) number. If that occurs, you can use the *Multiply Customer Loads* command to multiply all of the load values by a minus one (-1) to correct the issue. You will need to re-solve if you use this command.

The lowest pressure values should be at the end of the cul-de-sac, and at the end of the “main” street. For this example, the calculated values should be about 14.4 psig. Are the pressures OK? If not, a dimensional unit (most likely the *Load Units*) may not be correct.

## **Notes & Considerations**

---

- The model loads only represent the loads of the customers immediately attached to the system. We probably should consider whether additional loads need to be applied to account for future growth. These loads could be added as node *Base Load* values.
- When the new customers were added, their load was applied evenly - one-half to the To Node and one-half to the From Node of the supply main. This yields reasonably realistic results. However, if you wish to be conservative you might want to override the automatic node assignment and cause the customer loads to be applied to the downstream node of the one-way pipe segments.
- The supply pressure was given in this example. In reality this value would be derived from an analysis of the upstream system, or from field data collected from pressure recording charts or gauges. Further, for a complete analysis, the impact of the new system on the upstream system should be considered. Adding load to the upstream system will cause the upstream pressures to be reduced, and may negatively impact both the existing and new system's performance.
- In this example, an existing original DXF file was used to create a background image to trace over. We assumed that the drawing was to scale and our check indicated that it was. What if it wasn't? Ideally we would go back to the original drawing and re-scale it in the CAD system used to create the DXF. This would ensure that the next time we used the file it would be to scale. If we didn't have a way of re-scaling the DXF file, we could rescale it in GASWorkS by setting an appropriate *Scale Factor* value in the Background Settings. Or, we could still use the file to trace over, and manually enter the correct pipe lengths when we entered the pipe data.
- When we created the new model we used the *New* item, set the Preference Settings, set the Graphic Settings, added a background image, then set the Default Data Values. If the desired settings were already made and previously saved in specific files, we could have used the *New Project* item to automate all of the steps just mentioned - specifying which default data values, preference and graphic settings, and background image files to use when the model is created.