

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.

Note - The following instructions are written from the perspective of using the new GDI style. The classic command icons are shown for those who may be using the Classic GDI style.



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...

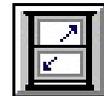
1) Create The New Model

- From the *File* menu, select the *New* item. The File Selection screen will be displayed. Enter a *File Name* for the new model, then click the *Save* command button. You can use either GASWorks 10 (.hdr) or GASWorks 11 (.g wz) file format.

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.

If the GDI Window is not automatically resized to the fit the GASWorks Application Window, resize the GDI Window using the *Maximize GDI Window* item from the *GDI Utilities* branch of the Command Tree.



2) Set Options & Settings

For this model we should set and check a number of settings before we get started building the model. Most settings and options are set using the *User Options & Settings* screen. To open the screen select the *Options & Settings* item from the *Utilities* menu. The User Options & Settings screen will be displayed.

Default Dimensional Units...

Select the *Units* data tab. Review the data tab to ensure that the following values are selected:

| Parameter Groups... | Dimensional Units... |
|---------------------------------------|----------------------|
| Coordinate Values | Feet |
| Customer Loads | Cfh |
| Diameter Values | Inches |
| Efficiency Values | Decimal |
| Elevation Values | Feet |
| Heating Value (Energy Content) Values | Btu/cf |
| Length Values | Feet |
| Node Load Values | Mcfh |
| Pipe Flow Values | Mcfh |
| Pressure Values | Psi |
| Prompted Length Values | Feet |
| Temperature Values | Fahrenheit |
| Velocity Values | Feet/sec |
| Viscosity Values | Lbm/Ft-sec |

Default Data Values...

Select the *Default Data* data tab. Review the data tab to ensure that the following values are set or selected:

Use Last Record As Default = **Selected**

| | | |
|----------------------|-----|------|
| Customer Load | 0 | Mcfh |
| Node Base Load | 0 | Mcfh |
| Node Base Load Known | Yes | |
| Node Elevation | 0 | Feet |
| Node Pressure | 0 | Psi |

| | | |
|----------------------------------|--------------|------------|
| Node (Gas) Temperature | 60 | Fahrenheit |
| Node Temperature Known | No | |
| Pipe Flow Equation | IGT-Improved | |
| Pipe Hydraulic Efficiency | 1 | Decimal |
| Pipe Size/Type (Inside Diameter) | 2P | Inches |
| Allow Pipe Sizing | No | |

The remaining items and settings on the tab should be acceptable.

Default Gas Properties...

Select the *Gas Properties* data tab. Review the data tab to ensure that the following values are set:

Basis Of Volume Measurement...

Base Pressure: Psi

Base Temperature: Fahrenheit

Default Gas Property Values...

Specific Gravity:

Viscosity: Lbm/Ft-sec

Heating Value: Btu/cf

Specific Heat Ratio:

Values Are Unknown

Environment Settings...

Select the *Environment* data tab. If the Prompt For Data During New Feature Entry is selected, unselect it.

The remaining items and settings on the tab should be acceptable.

Graphic Settings...

Select the *Graphics* data tab. Review the data tab to ensure that the following values are set or selected in the *Settings* section:

| | | |
|-------------------------------|----------------|--------------------|
| Customer Symbol Display Limit | 10000 | Feet |
| Customer Symbol Size | 2 | % Of Display Width |
| Customer Symbol Style | Circle - Solid | |
| Feature Snap Ratio | 3 | |
| Flow Arrow Display Limit | 10000 | Feet |
| Flow Arrow Style | Chevron | |
| Grip Symbol Size | 1 | % Of Display Width |
| Node Symbol Display Limit | 10000 | Feet |
| Node Symbol Size | 1 | % Of Display Width |
| Node Symbol Style | Circle - Solid | |

The remaining items and settings on the tab should be acceptable.

Click the *Close* command button to save and apply the changes.

4) Attach The Background Image

For this example we will attach a background image to model, then draw 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. Navigate to the folder (usually the *Files* folder of the root GASWorkS folder) containing the desired file - **plat.dxf**. Select the file, then click the *Open* command button.

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

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

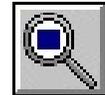
Note - The layers section contains the layers found in the DXF file. Individual layers can be turned “on” or “off” using this list. For now, ensure that all of the layers are turned on (selected).

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.

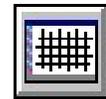
5) Zoom To Display The Background Image

- 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 visible in the GDI Display, click the *Display Background Image* item from the *Display Settings* branch of the Command Tree. The background image should now be displayed.



6) Check The Scale Of The Background Image

It is always good practice to check the scale of the background image before tracing over it. Let’s check the scale of this drawing.

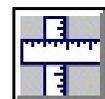
Zoom into a portion of the GDI Image where the right-of-way (ROW) can be measured.

- Use one of the Zoom methods to the area around “Golden Eagle Drive” so that the distance between “Lot 76” and “Lot 82” may be measured.



Measure the distance between the ROW lines.

- Select the *Measure Distance* item from the *GDI Utilities* branch of the Command Tree.
 - For the *First Point*, left-click near the west (left) ROW line.



- For the *Next Point*, left-click near the east (right) 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? If not there is an issue with the scale of the background image. Check the background image settings.
- Click the *OK* command button to clear the distance message.

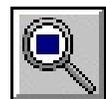
7) Turn “Off” Some DXF Layers

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.

Note - You may need to scroll the layer list left to find the PP-Text item.

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



8) 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.



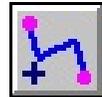
Note...

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...

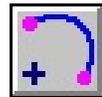
- Select the *Add Polyline Pipe* item from the *Construction* branch of the Command Tree.
 - For the *From Node Location*, select the desired location by left-clicking the mouse.
 - For the following vertices, continue to select the desired locations by left-clicking the mouse.
 - After the vertex representing 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. Select a location on the opposite side of the cul-de-sac for 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.

9) 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.



- Select the *Trace* item from the *GDI Utilities* branch of the Command Tree. 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?

If not correct the issue using the Pipe Tap command, or Move Pipe end command as appropriate.



If a correction was made, 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). 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.

When all corrections are made, click the *Zoom To Fit* icon from the lower-left corner of the GDI Window to display the entire model.



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

10) 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 data values as appropriate, then adding the remaining customers. The customer locations should look similar to Figure 1.

Add the first customer.

- Select the *Add Customer* item from the *Customer* branch of the Command Tree. 
- For the *Customer Location*, left-click in the center of one of the lots shown plat.
- For the *Supply Main*, left-click on the adjacent main (pipe).
- Left-click on the customer symbol to display the Customer Data in the Data Panel. In the *Hydraulic Data Items* section, set/check the following values:

| | |
|-----------------------|------------|
| Node Load Application | Both Nodes |
| Unit Count | 1 |
| Per Unit Load | -100 |
| Load Units | Cfh |
| Adjust Load | Yes |
| Load Status | On |

Note - The negative load value indicates that gas leaving the system.

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

Continue adding the rest of the customer features.

- Select the *Add Multiple Customers - Auto Assigned Main* item from the *Customer* branch of the Command Tree. 
- Select the location for each customer feature (add a customer for each full lot shown on the plat). 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.

11) Spot Check The Data

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

- Select the *Text Display Settings* item from the *Display Settings* branch of the Command Tree. The Text Display Settings screen will be displayed. On the *Customer Items* data tab, set the following values:



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

Customer Load Per Unit = **Select**

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

There should be a non-zero load value for every customer.

12) 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:

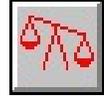
Pressure = **15**
Pressure Units = **Psi**
Pressure Known = **Yes**

Note - The *Base Load Known* will automatically be set to “No” and the value will be calculated when the model is solved.

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

13) 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.



14) Review The 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* item from the *Display Settings* branch of the Command Tree.
- Display the pressure values by selecting the *Text Display Settings* item from the *Display Settings* branch of the Command Tree. 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. Or, use the grip for the pipe text to move the text away from the pipe.

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 14.9 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 can 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 for multiple loads distributed along a main/pipe segment. You can change this setting to apply all of the customer to either the From node or To node of a pipe segment.
- 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, a drawing of a plat in DXF format was used to create a background image to trace over. We anticipated 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 previously saved default data values, preference and graphic settings, and background image files to use when the model is created. Alternatively, we could have specified a previously created and saved "template" model to establish the appropriate settings and default values.