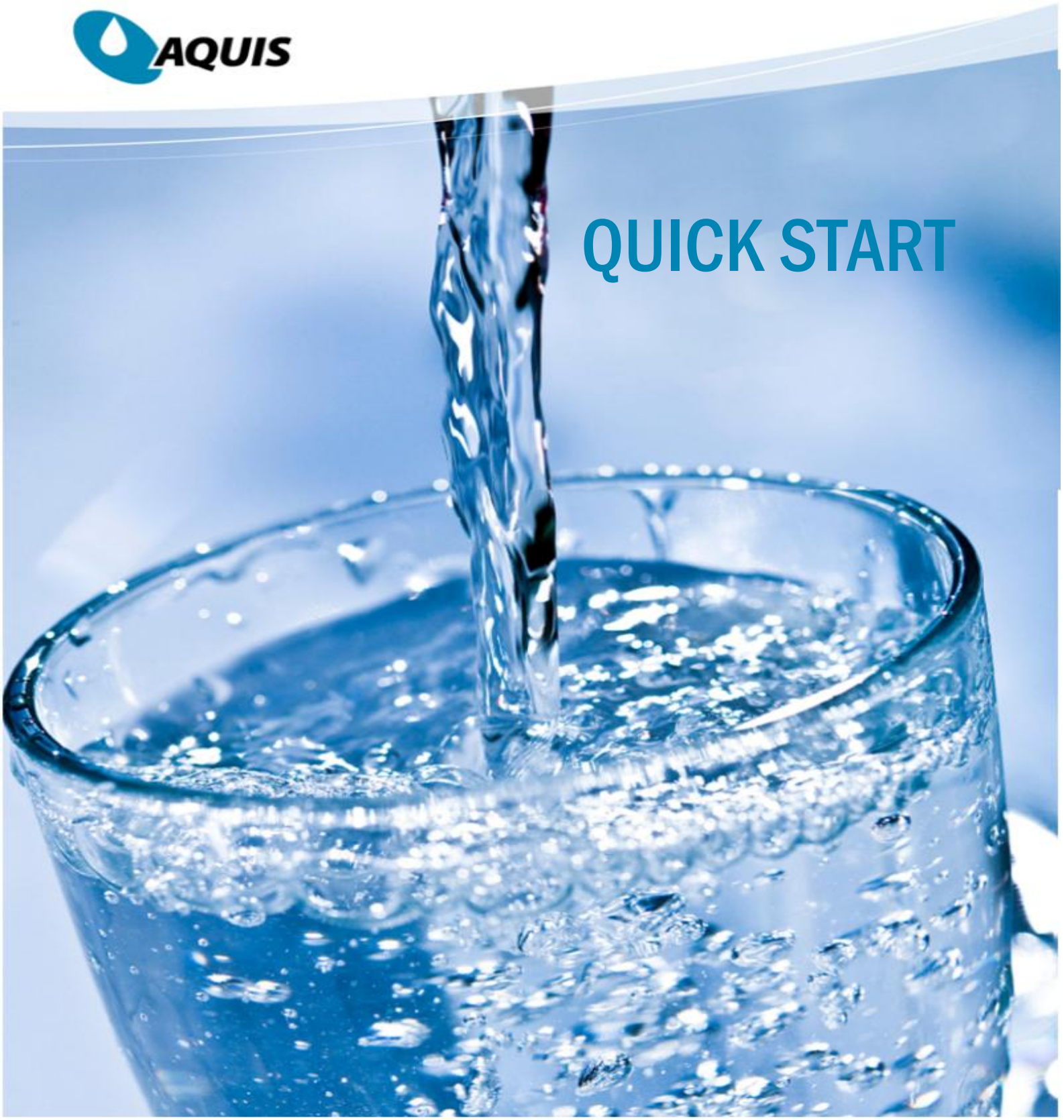
WATER NETWORK MANAGEMENT

Real-time monitoring and proactive decision making

PROVIDING CLEAN WATER TO MORE THAN 100 MILLION HOMES

AQUIS

QUICK START



QUICK START

AQUIS free1000

AQUIS free 1000
- Hydraulic Modeling -

Lesson 1

Lesson 2

Lesson 3

QUICK START

AQUIS free1000

Note:

Prior knowledge of hydraulics is not a prerequisite for taking these lessons, but it may help to put things into perspective. The objective of the lessons is to give you an idea of some of the features provided by AQUIS free 1000. You are assumed to have basic knowledge of your operating system.

Lesson 1: Introduction to AQUIS FREE 1000

OBJECTIVES

In this lesson you will

- Be introduced to the AQUIS free 1000 user interface and learn basic navigation.
- Learn about key objects such as nodes and pipes.

LESSON OVERVIEW

The introduction material to perform hydraulic modeling using AQUIS free 1000 consists of the following lessons. You are strongly recommended to study these lessons prior to working in AQUIS. The estimated time to complete the lesson is stated in parentheses.

Lesson # 1: Introduction to navigation and model creation (Approx. 50 minutes)

Lesson # 2: Simulation (Approx. 10 minutes)

Lesson # 3: Results (Approx: 25 minutes)

REFERENCES

You are recommended to study the documentation provided with the product. In the application you can use F1 on the keyboard to access context sensitive help, or you can launch the documentation from the Help menu.

7-Technologies encourages customer feedback. Each topic within the online Help contains a link to write the 7-Technologies' documentation department and suggest changes, point out flaws, etc.

LEGEND

	Read. The chapter contains background information about the Lesson subject. The contents introduce you to some of the concepts that you will meet when you take the exercise.
	Do. Step-by-step instructions to complete the exercise(s) for the Lesson subject.

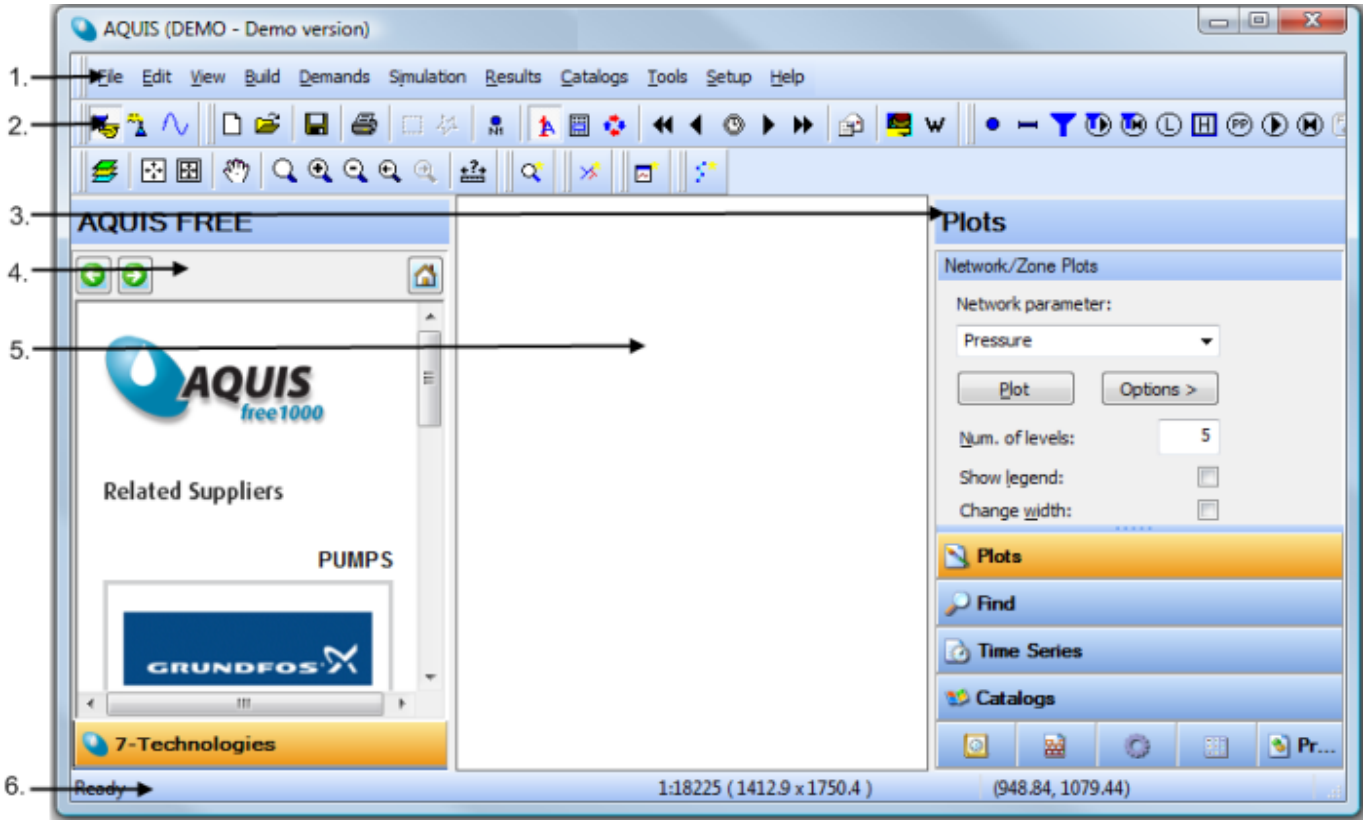
NAVIGATING THE AQUIS APPLICATION

If you have installed AQUIS to the default folder, you can access the application from the desktop icon or as follows:

- Running Windows Vista: **Start**>**All Programs**>**7T**>**AQUIS**.
- Running Windows XP and 2000: **Start**>**Programs**>**7T**>**AQUIS**.

User interface

The AQUIS user interface consists of the following overall elements:



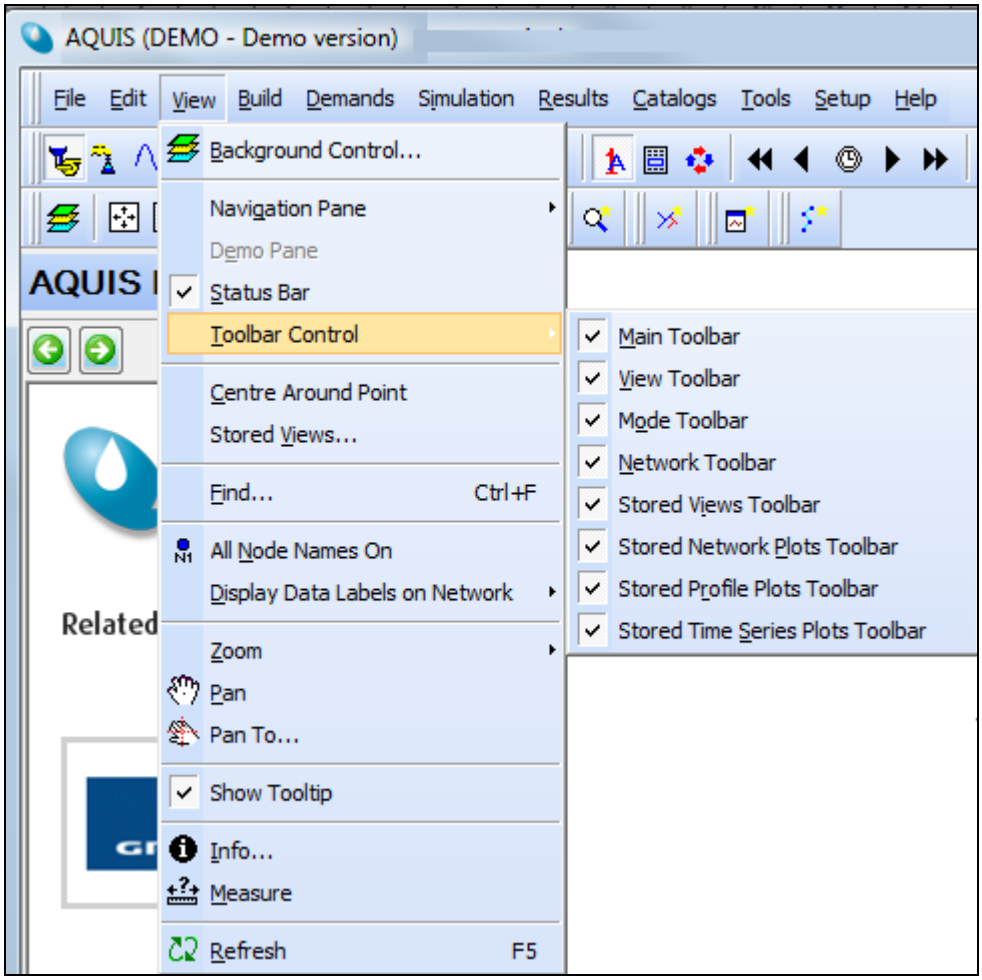
AQUIS user interface.

1. **Menu bar**.
2. **Toolbar**. Each toolbar button is a shortcut to an AQUIS command. For further details about the toolbar buttons, see the ensuing section on Toolbars.
3. **Navigation pane**. Access to core functionality. It launches a quick access to investigate, analyze and modify your model.
4. **Banner pane**. This section provides direct access to Web sites with related products, such as to manufacturers of pumps, valves, etc.
5. **Workspace**. Where your model will be displayed.
6. **Status bar**. It displays the current operational status. (When no operation is being performed, the status bar displays "Ready", but as various operations are carried out, the status message changes.)

Toolbars

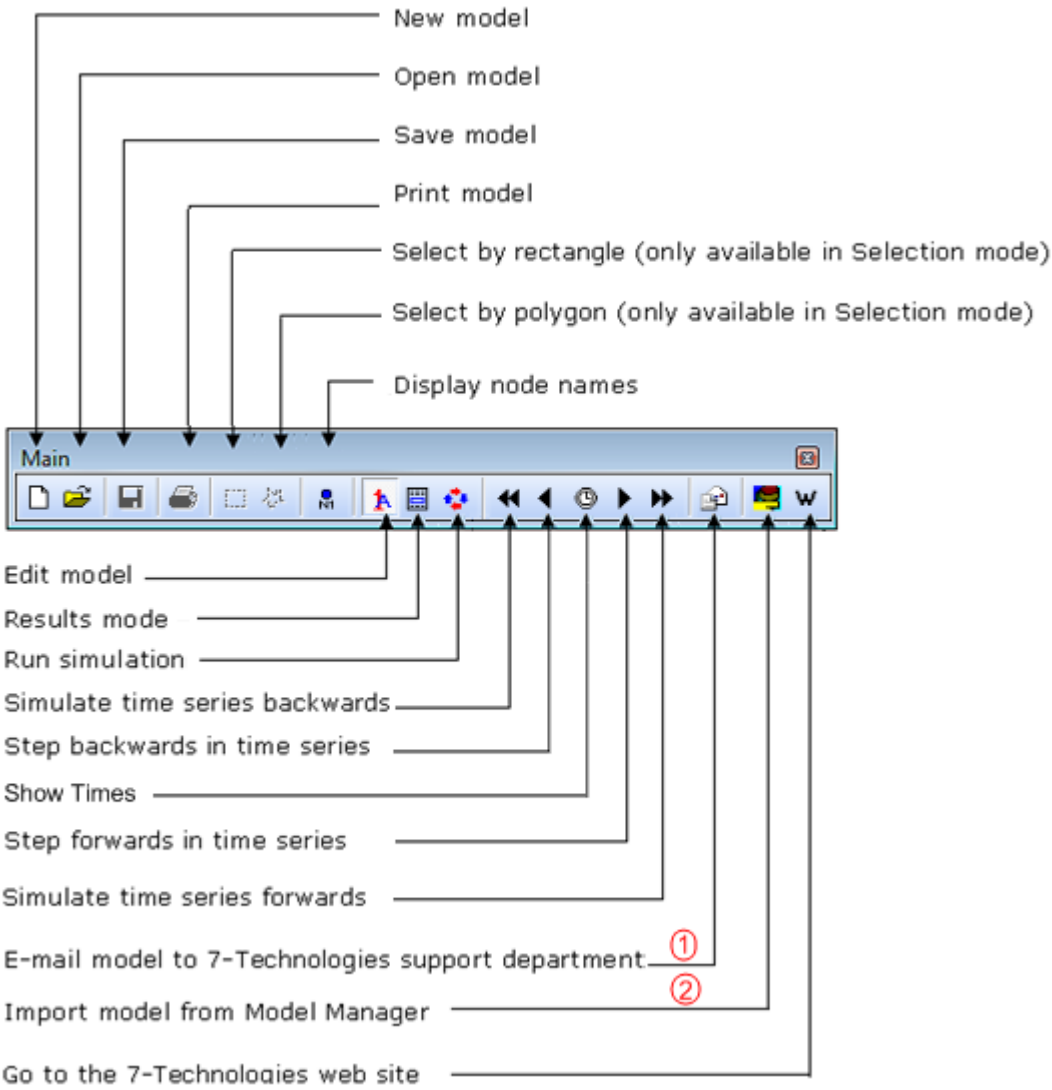
The AQUIS user interface provides you with a number of toolbars, which can be dragged to any position on the screen, within or even outside the AQUIS program window.

To define which toolbars to display, select **View**>**Toolbar Control**. Select or clear the relevant check boxes.



Toolbar Control menu

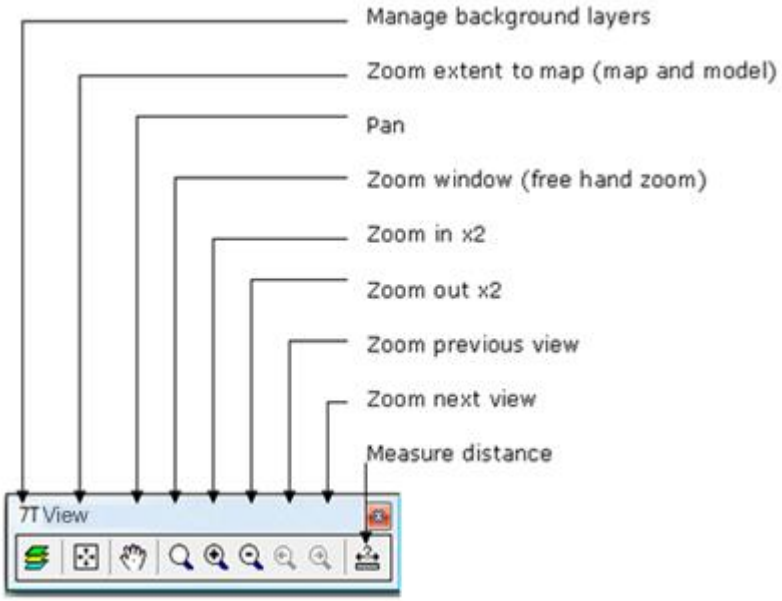
Main toolbar



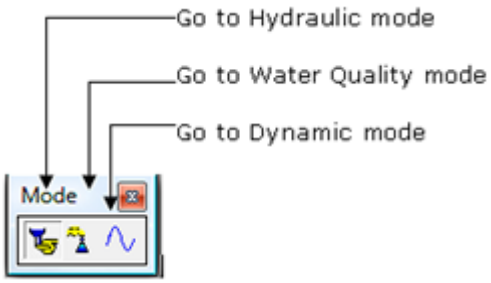
1) The e-mail address to 7-Technologies is for licensed customers only. However, your feedback is highly appreciated, but you cannot rely on receiving support services.

2) You will need a licensed version of Model Manager to use the import function.

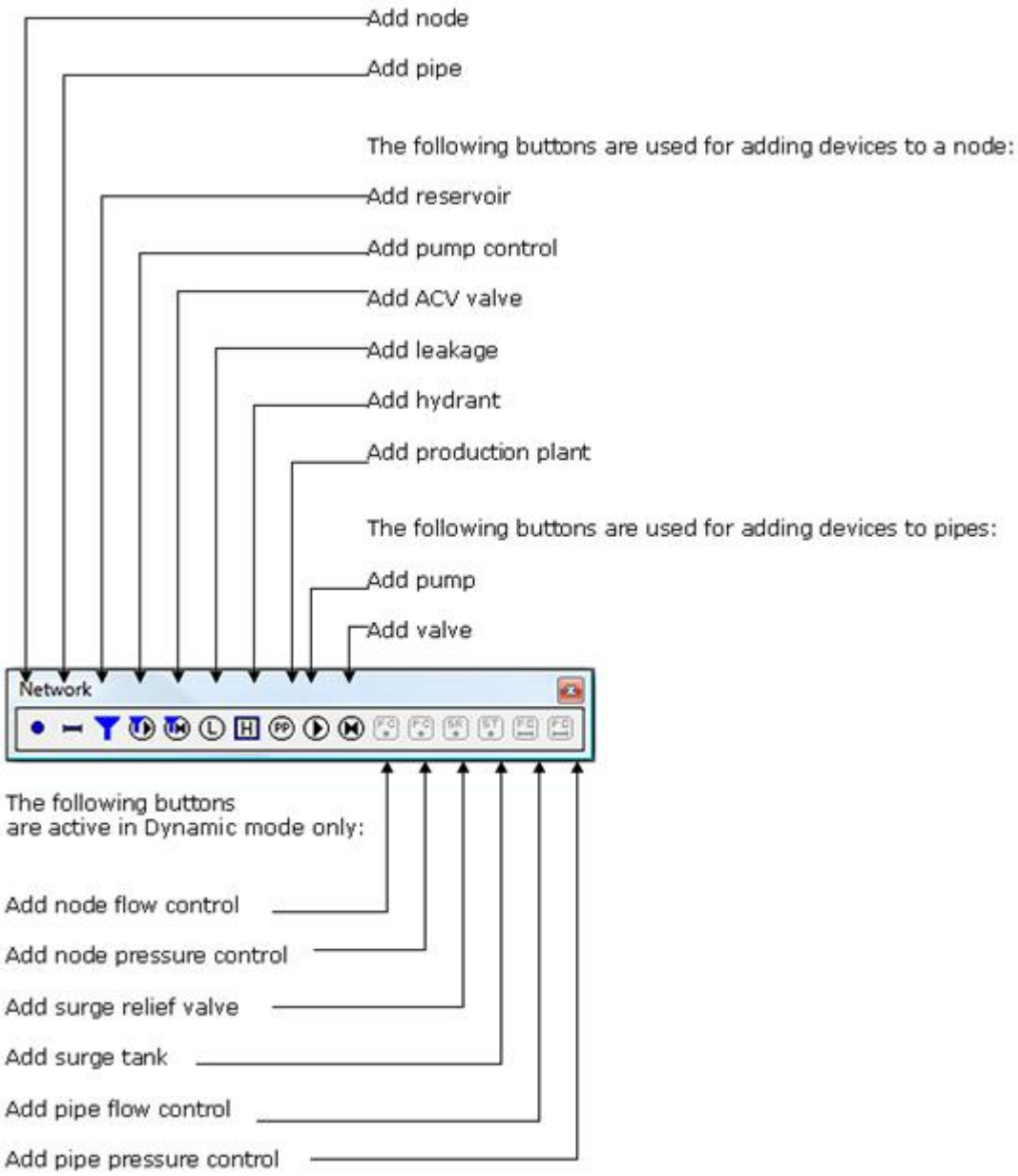
View toolbar



Mode toolbar

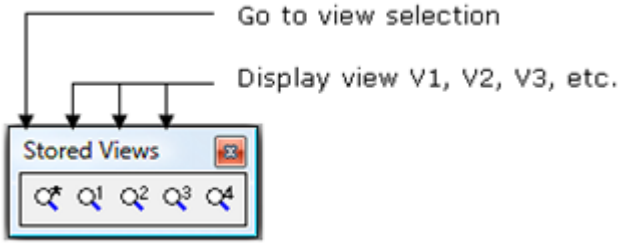


Network toolbar



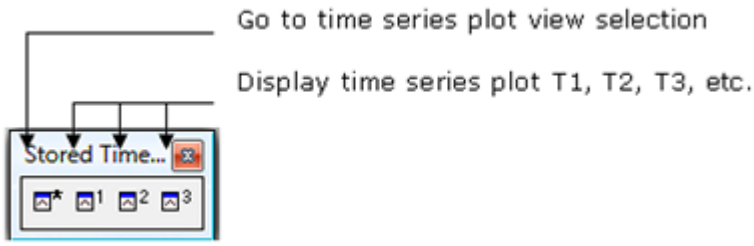
For the following 4 plots, the number of icons on the toolbar depends on how many plots you actually define.

Stored Views toolbar



Stored Profile Plots toolbar

Stored Time Series Plot toolbar

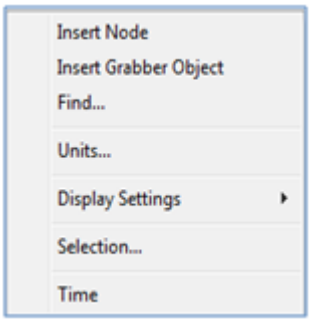


Stored Network Plots toolbar

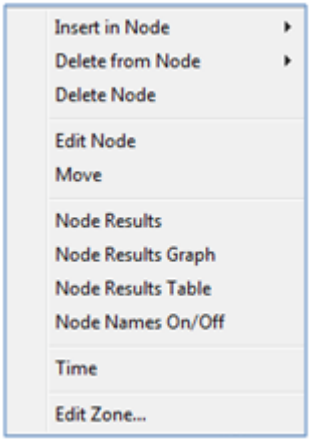
Right-click menus

When you right-click the various areas in the workspace you will get access to additional options and functionality. The menus are context dependent, which means that you will see different right-click menus display depending on where you right-click. You are recommended to investigate the right-click menu and in the following you will see some of the options once you have defined the appropriate objects.

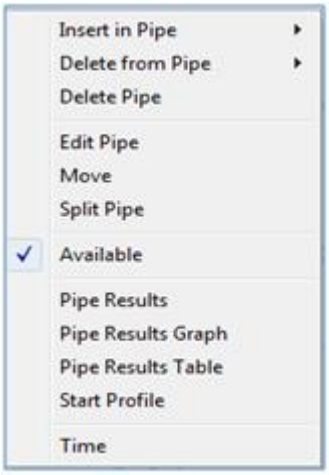
Right-click the workspace to view these options:



Right-click a node to view these options:



Right-click a pipe to view these options:



NODES, PIPES, AND BASIC DEMANDS

In AQUIS you define the demand flows and pressures on the nodes.

Boundary Conditions

The boundary conditions are prerequisite input data for the hydraulic operations performed by AQUIS. The boundary conditions are parameters that are defined at the node level.

These are mandatory settings.

An active boundary condition must be activated by selecting the check box for BC under the Node dialog box.

For details refer to the ensuing exercise section on how to create nodes.

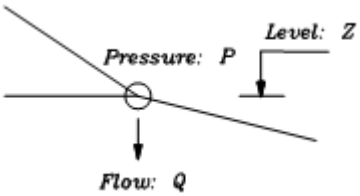
Most nodes will have the demand flow as a boundary condition (i.e. these are known) and the pressure at those nodes will be required to be calculated (unknown, thus it is not a boundary condition).

At the node that acts as the inlet to the system, the pressure will be the boundary condition and the flow into the system set as unknown.

Nodes

A node can be several consumers, a simple node without consumption, or a source (inlet).

The simplest node is a junction between pipes, which can have no consumption allocated at it.



It should be mentioned that different device types such as a Reservoir, Inlet pump, Altitude Control Valve (ACV), Leak and Hydrant can be inserted into a node. This is, however, not part of the scope for this lesson. If a device is allocated to a node, then the appropriate button in the Node dialog box will be active or an appropriate pane will appear. For in-depths information about the various devices and how they are configured, you are referred to the online Help provided with the application.

Pipes

Pipes are connections between nodes. Pipe characteristics include an upstream node, a downstream node, length, internal diameter, and roughness.

A pipe can have any type of valves or pumps associated. The device must, however, be located at one of the pipe ends and there can only be one device allocated to each pipe. If a device is attached to a pipe, then the dialog box will display the appropriate tab page for the device.

EXERCISE

This exercise shows how to create two nodes and a pipe. To get started, however, you first need to create a model.

Creating a model

In a real-life scenario, you will not create your model from scratch, but use digital GIS data from sources such as the Model Manager. This is, however, beyond the scope of this training exercise. You are recommended to study the description provided in the online Help for importing data.

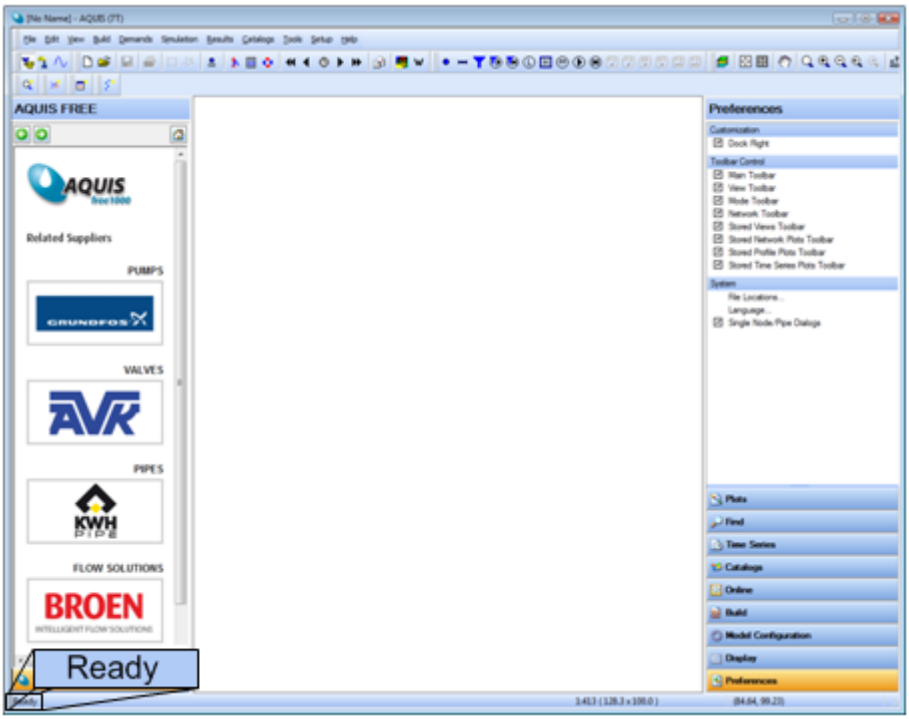
For training purposes, this exercise will take you through the basic steps of creating a model, adding nodes and pipes.

Throughout the exercise you will be provided with the necessary names and values for the parameters and attributes. If you use the provided names and values you can easily compare your result with the results displayed in the screen captures in this exercise.

To create a model

- From the menu bar select **File**>**New** or click the icon on the toolbar.

  In the bottom left side of the status bar you can see the message Ready. See the figure in the following.



You are now ready to start adding components to your model as described in the ensuing sections.

Adding a node

You will create two nodes. To create the nodes you are recommended to use the following information:

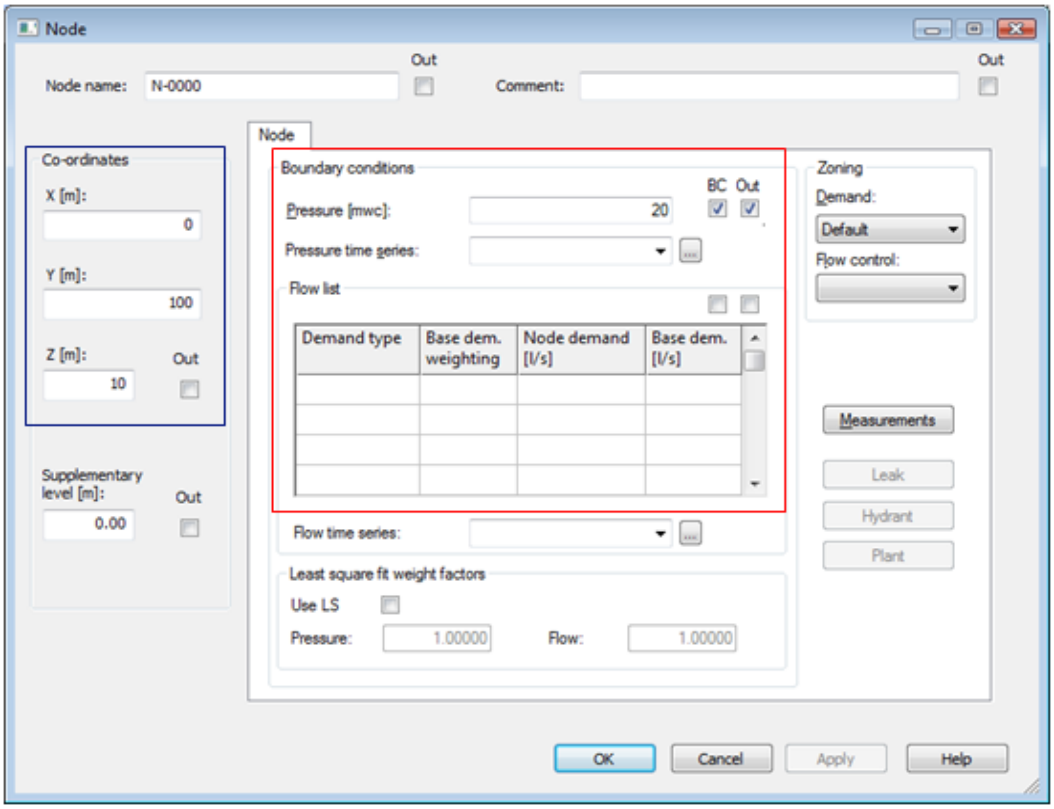
Name	X[m]	Y[m]	Z[m]	Pres-sure (mwc)	Demand Type	Node Demand	BC Pressure	BC (flow)	Comment
N-0000	0	100	10	20	Blank	Blank	Select checkbox	Clear checkbox	Node 1. The inlet node.
N-0001	1000	-1000	15	*	1	0.5	Clear checkbox	Select checkbox	Node 2.

Notice that the asterisk (*) indicates that the attribute fields is included in the calculation.

To create nodes

1. From the menu bar select **Build**>**Add**>**Node** or click the icon on the toolbar.
2. Left-click anywhere in the workspace. (In the following steps you will see how the coordinates determine the node position).

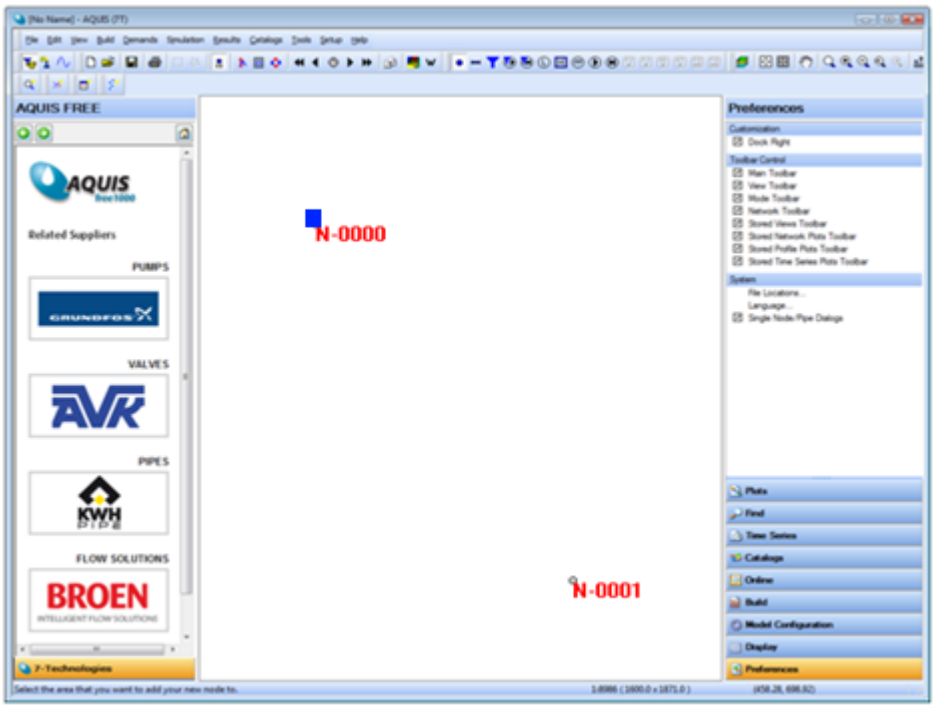
   This opens the Node dialog box. If the dialog box does not open, right-click the node and select **Edit Node**. See the figure in the following.



3. Under **Node name** enter **N-0000**.
4. Under **Co-ordinates** enter the value for **X**: **0**.
5. Enter the value for **Y: 100**.
6. Enter the value for **Z**: **10**.
7. For the boundary conditions enter the value for **Pressure**: **20**.
8. Select the checkbox for **BC** (**pressure**).
9. For **Demand type** under the flow list table leave the entry field blank.
10. For **Node demand** in the same table leave the entry field blank.
11. Ensure that the checkbox for **BC** (**flow list**) is cleared.
12. Leave the rest of the entry fields as default.
13. Press **OK**.

    If you fail to see the node, you can click the icon Zoom model extent on the toolbar to get a big picture of the model.
14. Now add the second node. Follow the preceding steps (1 through 13), but ensure to enter the correct values (as specified in the table) for the second node, **N-0001**.
15. To see the node names select **View**>**All Nodes Name On**, or click the icon on the toolbar.

    If you followed the guidelines in the preceding your model looks like the following figure.



Saving the model

You are recommended to save your model at this point in time. It is, however, also possible to do it after you have connected the nodes to the pipe.

To save the model

1. From the menu bar select **File**>**Save As.**
2. Select the appropriate folder for your model and enter the model name **FREE_1**.

Adding a pipe

When you have added your two nodes, you must insert a pipe to connect the nodes.

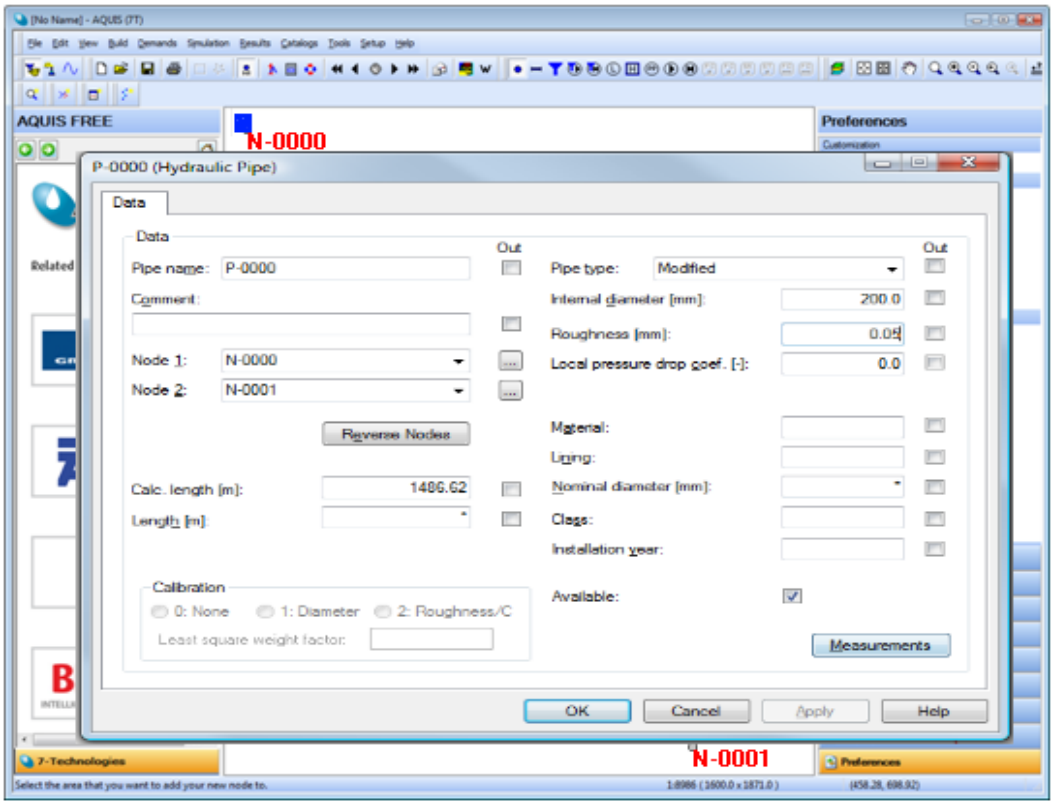
In this exercise you will be connecting P-0000 with the two nodes N-0000 and N-0001.

To create the pipe you are recommended to use the following information:

Name	Ups node	Dws node	Int. diam [mm]	Roughness [mm]	Local pressure drop coef.
P-0000	N-0000	N-0001	200	0.05	0.0

To add a pipe

1. Select **Build**>**Add**>**Pipe** or press the icon on the toolbar.
2. Point to the node **N-0000** as the upstream node and then point to the node **N-0001** as the downstream node. This will open the Hydraulic Pipe dialog box. If the dialog box does not open, right-click the node and select **Edit Pipe**. See the figure.



3. Verify that the

   Pipe name is **P-0000**. The name is generated automatically.

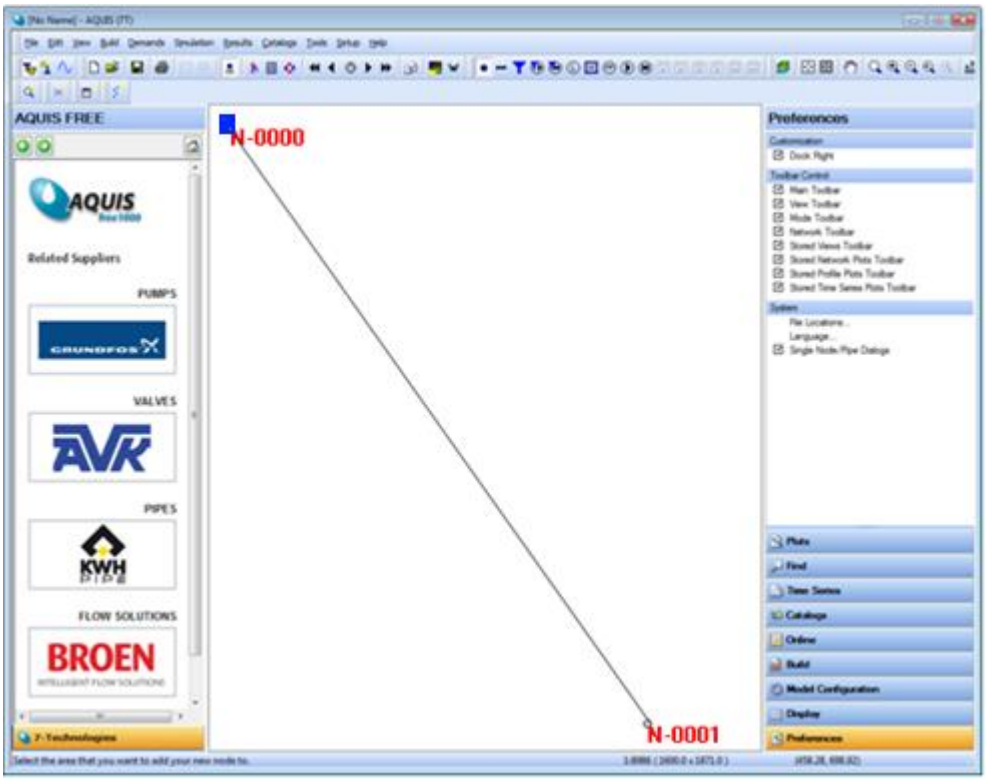
   Node 1 is **N-0000**.

   Node 2 is **N-0001**.

4. Under **Internal diameter** enter the value **200**.
5. Under **Roughness** enter the value **0.05**.
6. Under **Local pressure drop coef.** enter the value 0.0.

   This allows a pressure drop (representing minor losses from elbows, etc.) to be imposed on the pipe.
7. Leave the remaining options as default.
8. Press **OK**.

   If you have followed the instructions in the preceding steps your model will look like the figure in the following.



9. Save the model and proceed to the next section.

Adding a second model

You need to create a second new model using FREE_1 as the base. The second model is used in Lesson 2.

Use the guidelines in the preceding steps to create the second model. The specifications for the additional elements are listed in the following table.

Model element	Description
Model name	The name of the second model is **FREE_2**.
Nodes	Create three nodes **N-0002** **N-0003** **N-0004** See the node input in the ensuing section.
Pipe	Create three pipes. **P-0001** **P-0002** **P-0003** See the pipe input in the ensuing section.

To create a second model

1. From the menu bar select **File**>**Open** and locate the first model, **FREE_1**.
2. Select **File**>**Save As** and enter the name **FREE_2**.
3. Create the three new nodes.

Node specifications

Name	X[m]	Y[m]	Z[m]	Pres-sure (mwc)	Demand Type	Node Demand [l/s]	BC Pressure	BC (flow)
N-0002	2500	-1000	10	*	1	1.5	Clear checkbox	Select checkbox
N-0003	1500	-500	25	*	1	0.5	Clear checkbox	Select checkbox
N-0004	2500	-500	20	*	1	1.0	Clear checkbox	Select checkbox

In this example you should notice that there is only one node with a defined pressure, the remaining nodes are all flow nodes. This is just one very simple scenario. For models that require more than one pressure node, you are recommended to peruse the documentation provided with the application.

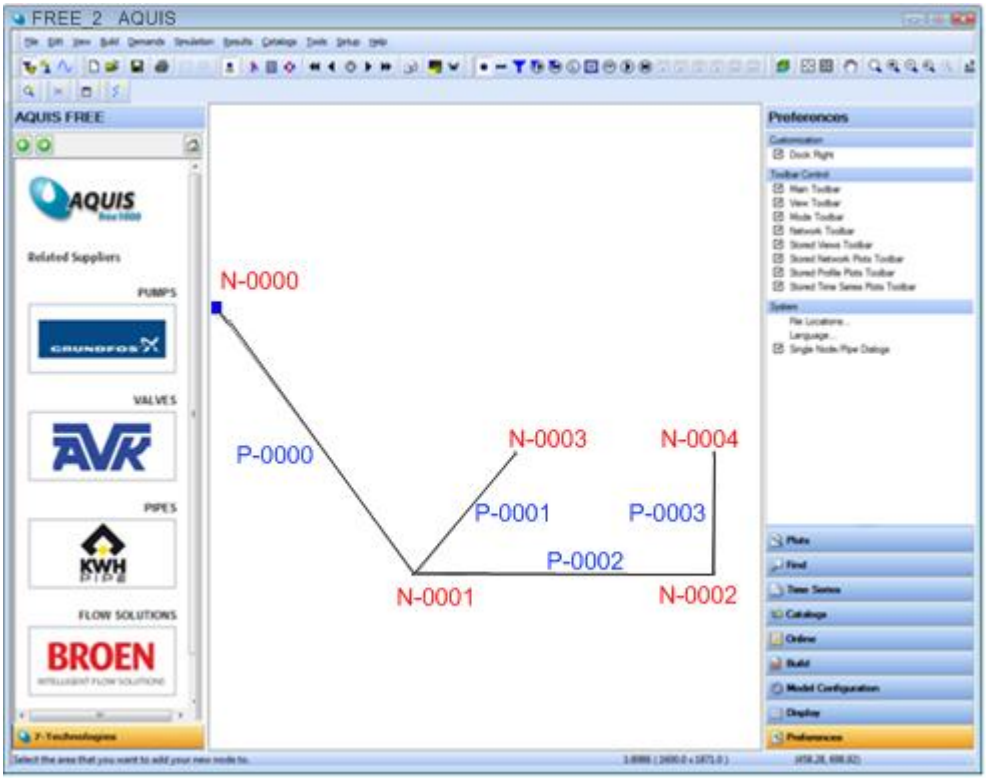
4. Create the three new pipes.

Pipe specifications

Name	Ups node	Dws node	Int. diam [mm]	Roughness [mm]	Local pressure drop coef.
P-0001	N-0001	N-0003	150	0.05	0.0
P-0002	N-0001	N-0002	100	0.05	0.0
P-0003	N-0002	N-0004	75	0.05	0.0

5. Save the model.

   If you followed the guidelines in the preceding your model now looks like this figure, without the labels for the pipe names. This view cannot be saved so when you open the model the next time, you will need to activate the display of the node names.



CONCLUSION

This lesson showed you how to move around in AQUIS free 1000 and gave a high-level introduction to the basic functionality.

If you followed the guidelines in this lesson you have now successfully created two models: FREE_1 and FREE_2.

FREE_1 has two nodes and one pipe. FREE_2 is an extension of the first model and has 5 nodes and 4 pipes.

To view the status of the hydraulic operations in the model, however, you will need to run a simulation.

Proceed to Lesson 2 that will explain how to run a simulation.

Lesson 2: Simulation

OBJECTIVES

In this lesson you will

- Learn how to perform a simulation based on the values defined for the model components.

LESSON OVERVIEW

The introduction material to perform hydraulic modeling using AQUIS free 1000 consists of the following lessons. You are strongly recommended to study these lessons prior to working in AQUIS. The estimated time to complete the lesson is stated in parentheses.

Lesson # 1: Introduction to navigation and model creation (Approx. 50 minutes)

Lesson # 2: Simulation (Approx. 10 minutes)

Lesson # 3: Results (Approx: 25 minutes)

REFERENCES

You are recommended to study the documentation provided with the product. In the application you can use F1 on the keyboard to access context sensitive help, or you can launch the documentation from the Help menu.

7-Technologies encourages customer feedback. Each topic within the online Help contains a link to write the 7-Technologies' documentation department and suggest changes, point out flaws, etc.

LEGEND

	Read. The chapter contains background information about the Lesson subject. The contents introduce you to some of the concepts that you will meet when you take the exercise.
	Do. Step-by-step instructions to complete the exercise(s) for the Lesson subject.

HYDRAULIC SIMULATION

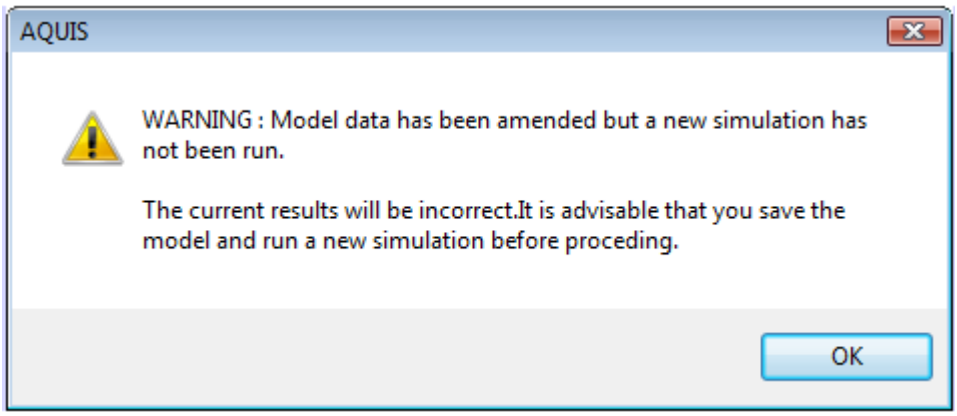
The objective is to analyze the hydraulic behavior of a water supply system. Simulation focuses on identifying flow, pressure, and velocity.

What this means in reality is that the model allows you to simulate and monitor the changes over time in your network and identify the hydraulic impact of the various elements in your model.

When you have built the model as explained in Lesson 1 and saved it with a valid file name, you are ready to run an offline simulation. The exercise in the ensuing will take you through the few steps of running a simulation.

If the model has changed and been saved without running a simulation you will experience the following:

- All elements (except Hydrants) will have a gray color.
- If the Output options are accessed you will see this message:



If for some reason the simulation cannot find a result, the system will generate a CHK file. The reason why you may not be able to run a simulation can for example be an incorrect definition of boundary conditions. In most cases the CHK file provides sufficient information to find the object or the area with the problem. When the situation is corrected you will need to run a new simulation.

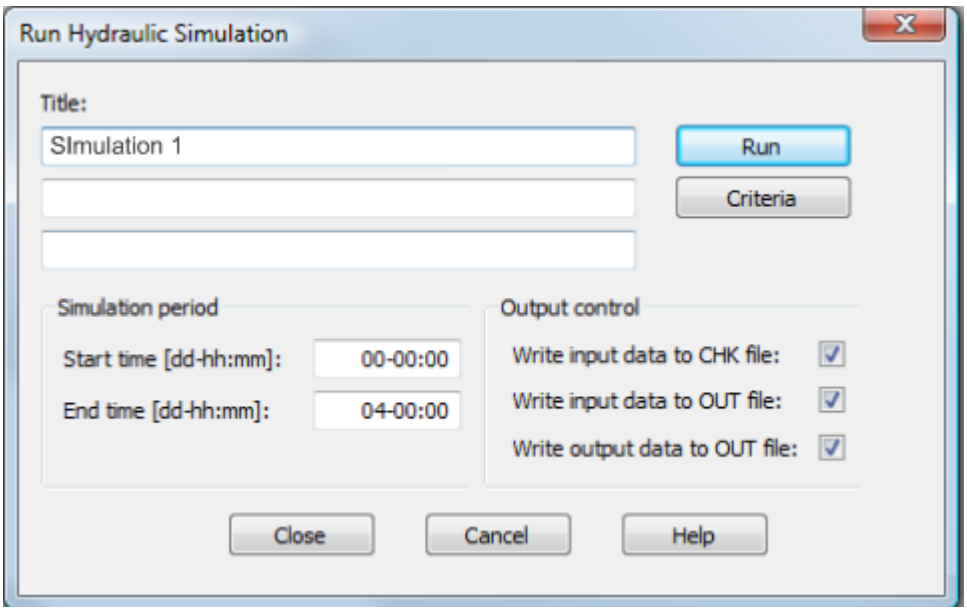
When you have completed the simulation, AQUIS automatically switches to Result mode. When you select an object the result dialog box will display. For details see Lesson 3: Results.

EXERCISE

This exercise explains how to perform a basic simulation based on the values defined in Lesson 1 for the model FREE_1.

To perform a simulation

1. Open the model **FREE_1** that you created in Lesson 1.
2. From the menu bar select **Simulation**>**Offline**, or click the icon on the toolbar. You will see the following menu.

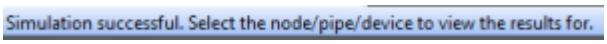


3. Configure the dialog box following these guidelines.

Simulation option	Description
Title	Enter **Simulation_1**. The title is transferred to the output.
Simulation period	Leave the Start time as default: 0-00:00 Set the End time to 04-00:00. The simulation will generate results for the complete period from midnight day 0 to midnight 4 days later.
Output control	This limits the amount of output written to the ASCII files. For large models you are recommended to limit the amount of output files for the simulation. For details on Results see Lesson 3. However, as the demo model is fairly small we will go all the way here.
	Select the checkbox for **Write input data to CHK file**.

Simulation option	Description
	Using this facility will result in identification of devices which are having problems being read into the engine.
	Select the checkbox for **Write input data to OUT file**. This will write out all the static input data to the OUT file, before the results.
	Select the checkbox for **Write output data to OUT file**. This will write out the output of the simulation to a text file after each time step.
Criteria	Leave the settings as default as this falls beyond the scope of this lesson.

4. Click **Run** to perform a simulation.
5. Ensure that the simulation is successful. If successful, there is a status in the lower left corner of the application.

   if the simulation fails you will be prompted with a message that states which correction measures you need to take.
6. Now run the simulation for the model **FREE_2**. Follow the preceding steps (1 through 5).

   Use the title name **Simulation_2** and apply the same settings as for the simulation of FREE_1.

CONCLUSION

This lesson showed you how to perform a basic simulation.

If you followed the guidelines in this lesson you have now successfully configured two models and run a simulation for both models.

Proceed to Lesson 3 where you will learn how to view the results of the simulation.

Lesson 3: Results

OBJECTIVES

In this lesson you will

- Learn how to view the results of the simulation of your model.

LESSON OVERVIEW

The introduction material to perform hydraulic modeling using AQUIS free1000 consists of the following lessons. You are strongly recommended to study these lessons prior to working in AQUIS. The estimated time to complete the lesson is stated in parentheses.

Lesson # 1: Introduction to navigation and model creation (Approx. 50 minutes)

Lesson # 2: Simulation (Approx. 10 minutes)

Lesson # 3: Results (Approx: 25 minutes)

REFERENCES

You are recommended to study the documentation provided with the product. In the application you can use F1 on the keyboard to access context sensitive help, or you can launch the documentation from the Help menu.

7-Technologies encourages customer feedback. Each topic within the online Help contains a link to write the 7-Technologies' documentation department and suggest changes, point out flaws, etc.

LEGEND

	Read. The chapter contains background information about the Lesson subject. The contents introduce you to some of the concepts that you will meet when you take the exercise.
	Do. Step-by-step instructions to complete the exercise(s) for the Lesson subject.

RESULTS

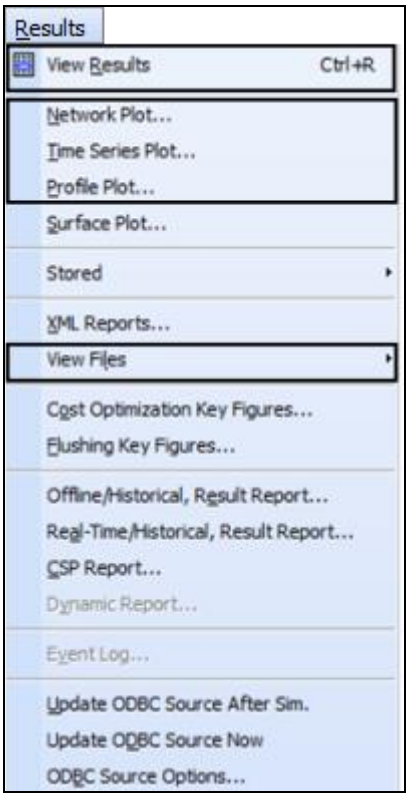
When you have performed a successful simulation as described in Lesson 2, you can view the simulation results for the nodes and pipes from the Results menu that you access from the menu bar. AQUIS free 1000 provides you with result options such as plot, dialogs, files, etc., to provide an overview of the hydraulic behavior of your water network.

The Results menu contains a number of result options that you can explore to ensure that your network is operating as expected.

This lesson only describes a few of the most commonly applied result options. You are recommended to explore the other options on your own.

In this lesson you will learn about the following result options:

- View results on nodes and pipes (**View Results**)
- View results based on plots (**Network**, **Time Series**, and **Profile** plots)
- View result files (**View Files**)



The ensuing exercises provide details on how to read the simulation results.

EXERCISE

This exercise explains how to view the results from the simulation performed as explained in Lesson 2.

Before you proceed, ensure that you have run a simulation for the model FREE_2.

The exercise will explain the following simulation results

- View Results
- Plots
- View Files

Option: View Results

When you have run a simulation you can view the results for the nodes and pipes.

To study the View Results option

1. Open the model **FREE_2**.
2. On the toolbar you will notice that the icon for **Results**  is activated. If this is not the case press **Ctrl+R** to activate the function.

   You can now view the results for the nodes and pipes.
3. Open the node **N-0000**. The simulation results appear on a separate tab page named **Results**. On this page you can see the results for the following parameters:

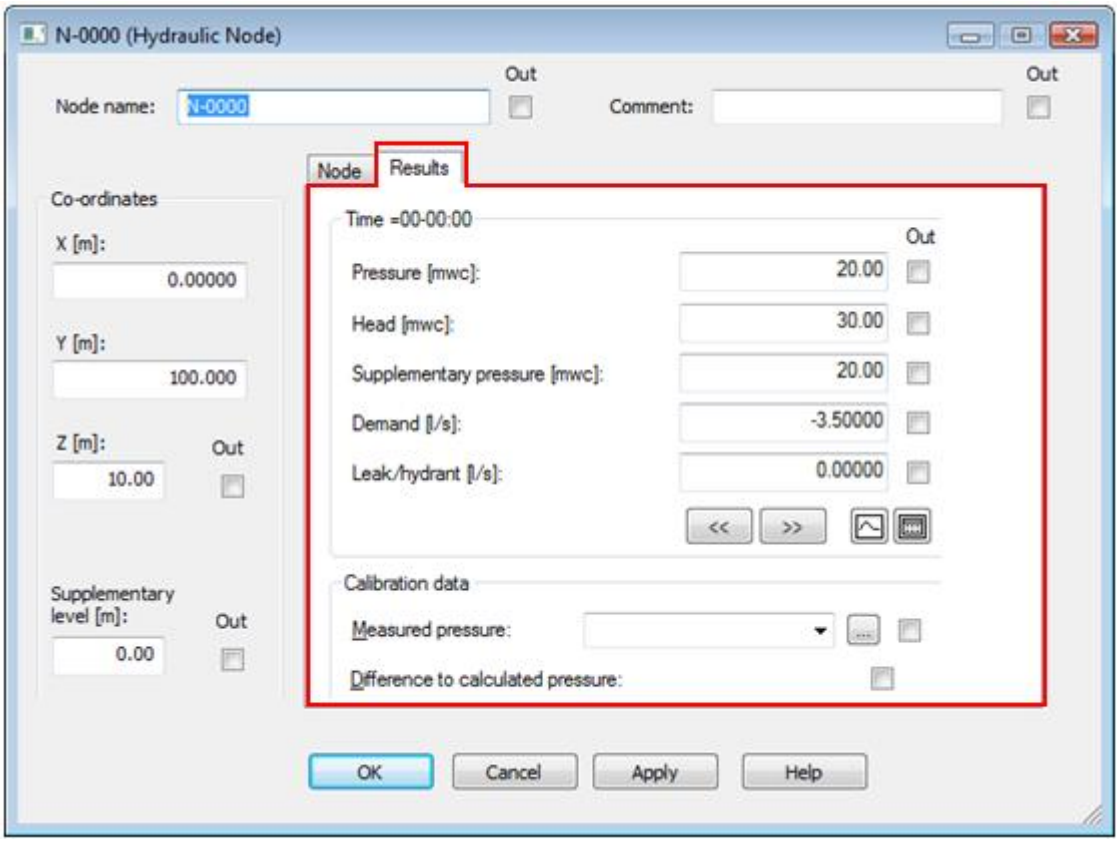
   - Pressure

   - Head

   - Supplementary pressure (that is the pressure at the highest property level)

   - Demand (Demand flow)

   - Leak/hydrant (flow)



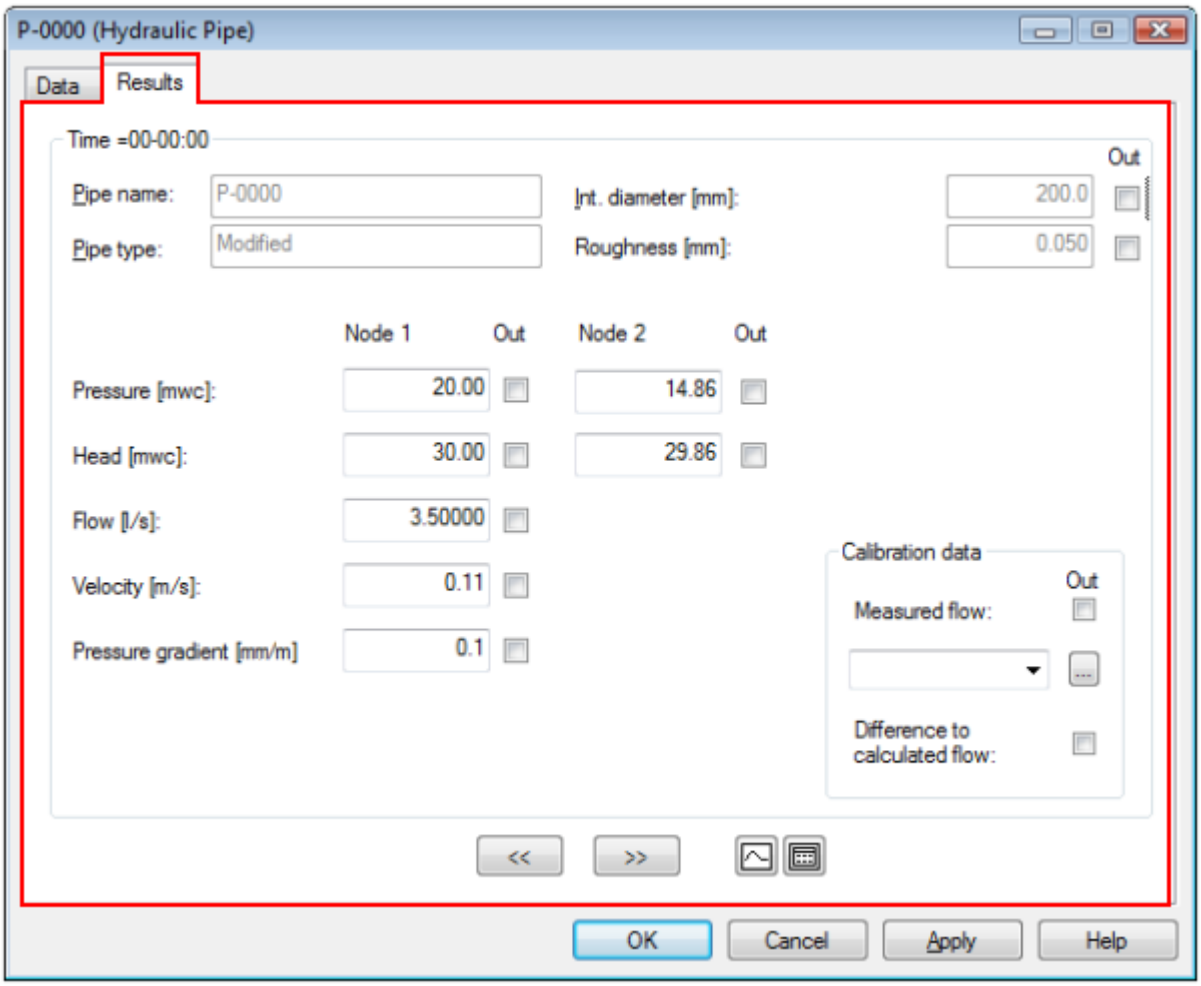
This node is a very simple node with no additional devices associated, such as pumps or valves. However, if there had been any devices on the node, you would have seen the results from these elements as well. Similarly for larger models, you can use the arrows << and >> to move forward or backward in time too see varying simulation results. In Lesson 2, you set the simulation period to 4 days, but since this is a very small model you will not notice any changes over time.

Notice that a positive flow is a demand on the system; while a negative flow is an input to the system. For this model, you can see that for the node N-0000 the Demand result is **-**3.5 l/s.

4. Now study the results for pipes. Open the pipe **P-0000**.

   On the Results tab page for the pipe you can view the results for the following parameters:

   - Pressure
   - Head
   - Flow
   - Velocity
   - Pressure gradient



This model represents the ideal world, but if for example you notice a model with a negative flow and negative velocity it shows that the flow runs in the opposite direction of how the pipe was configured.

Option: Plots

Plots present you with a graphical display of the simulation results.

In the following you will learn about the following plots:

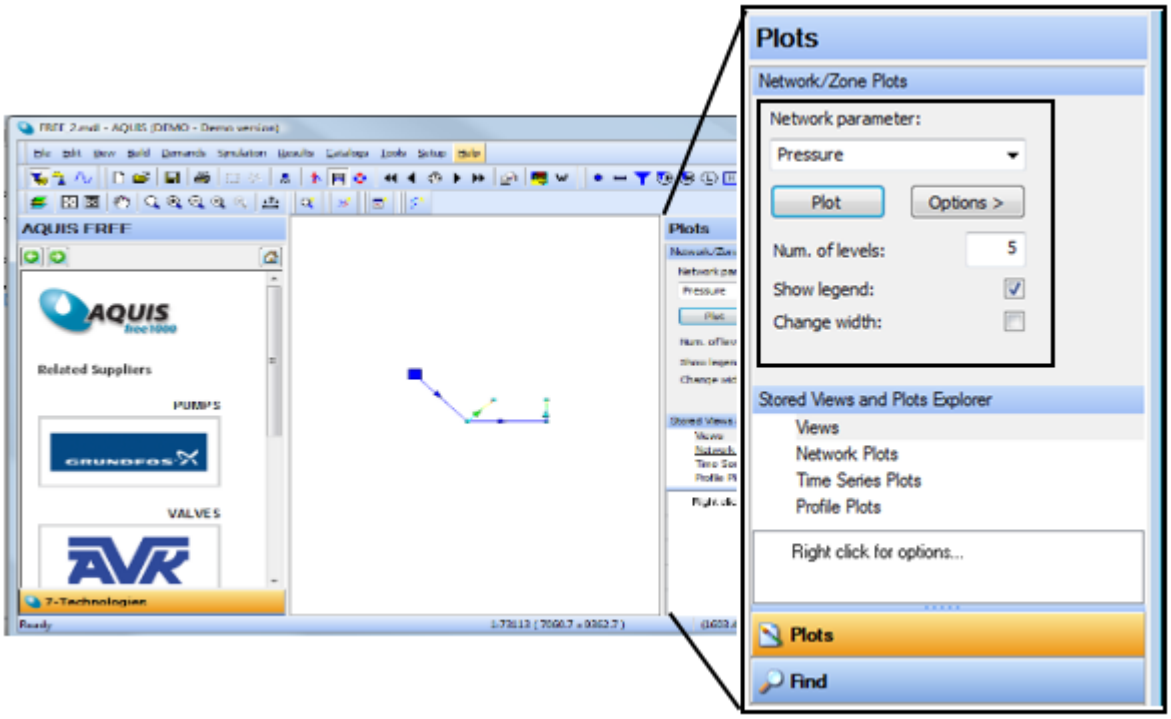
- Network plot
- Profile plot
- Time series plot

  The time series plot extends beyond the scope of this lesson. However, it is included since it represents the core area to monitor if your model includes dynamic behavior of daily demand profiles.

Network plot

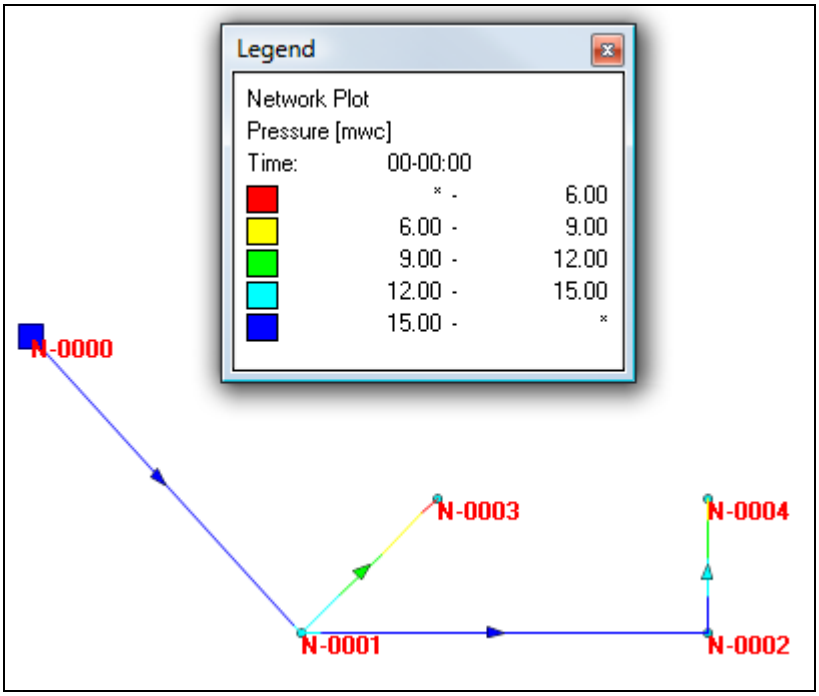
1. From the menu bare select **Results** and then **Network Plot**.

   This activates the Plots tab on the Navigation Pane to the right. By default the Plots tab is activated.



2. Under **Network parameter** select **Pressure** from the list.
3. Under **Num. of levels** enter the value **5**.

   This entry determines the number of intervals to display for the selected parameter. The minimum number allowed is 3 and the maximum is 12.
4. Select the checkbox for **Show legend** to show the legend in the work are for the model.
5. Press the button **Plot** to view the result.



Optionally you can

- Set plots for additional parameters.
- Customize the look and feel of the plot.

Parameters

You can try to set the plot for these parameters

- **Head**: The value of the total Head and flow direction.
- **Flow**: The flow direction and quantity.
- **Flow Changes (direction)**: The number of times that the flow changes direction during the simulation period.
- **Velocity**: Displays the velocity and direction of flow throughout the network.

Customize the look and feel

- Change color.

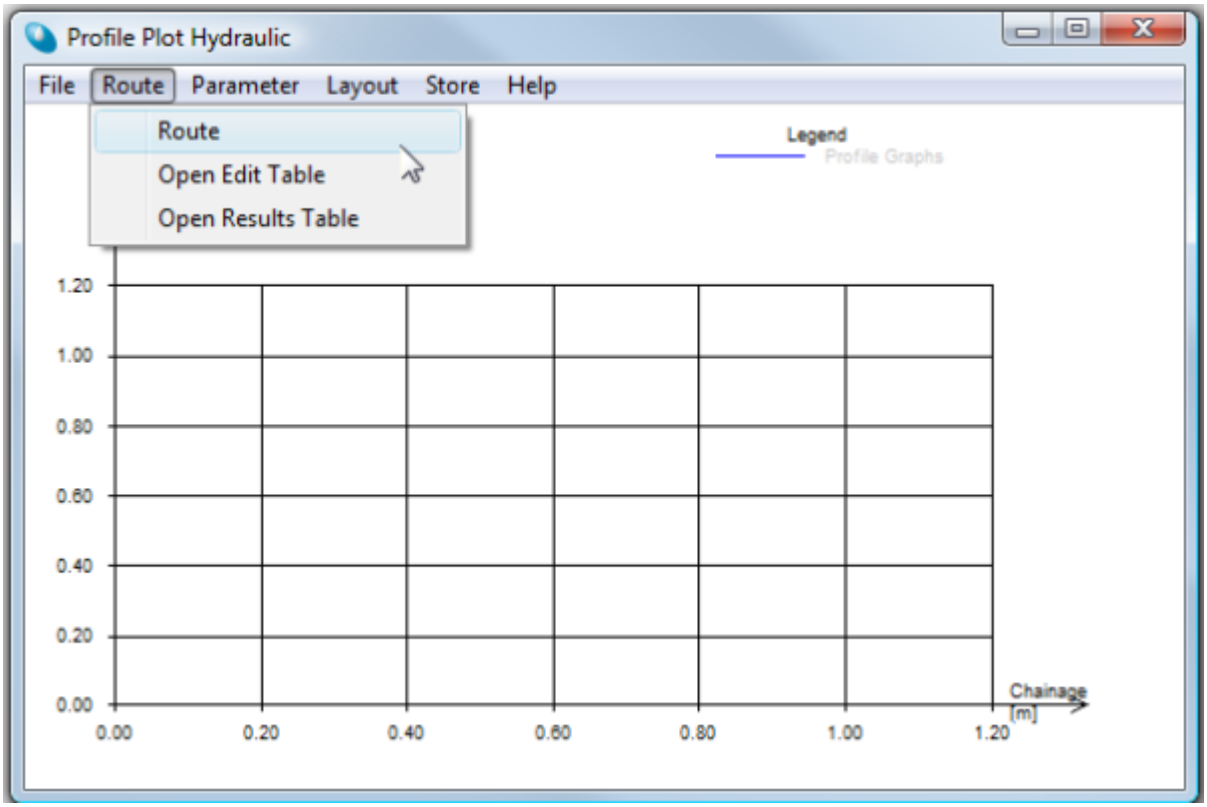
  Click the color to change the color scale.
- Change arrows, line width, etc.

  From the menu bar select Setup, then Display Settings, and point to Network Display.

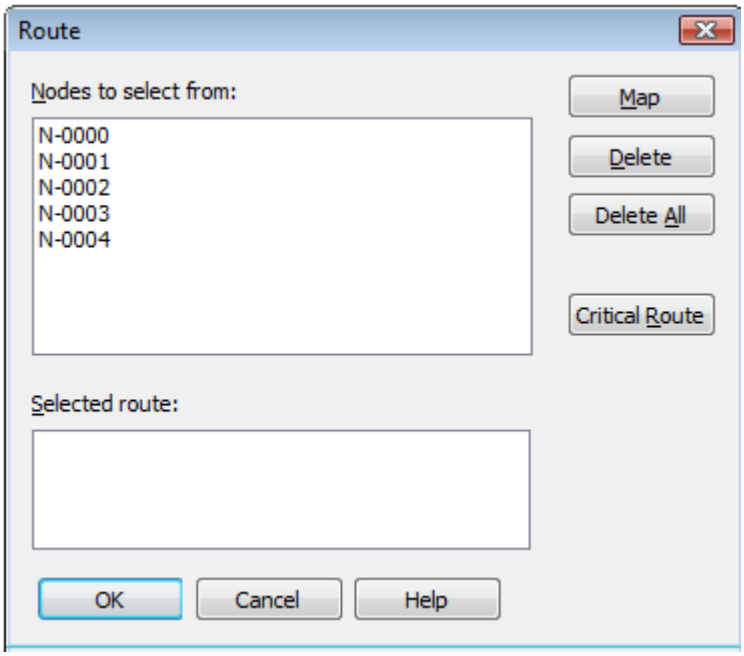
Profile plot

A Profile plot shows how parameters vary over distance.

1. From the menu bar select **Results** and then **Profile Plot** to open this dialog.
2. In this dialog box select **Route** and then **Route**.



3. Under the Route dialog box click the **Map** button.



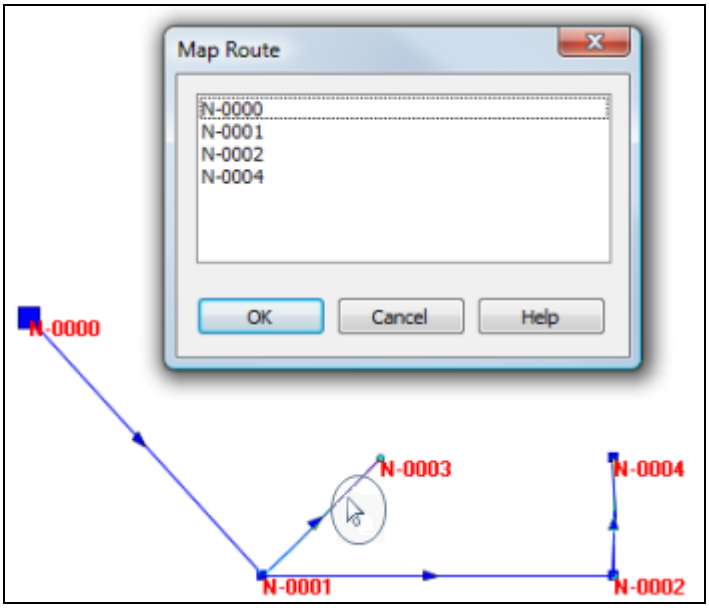
4. Under the **Map Route** dialog box double-click the following pipes in the model (this is illustrated in the following figure - notice the cursor):

P-0000

P-0001

P-0002

Notice how the associated nodes will be listed in the Map Route dialog box.

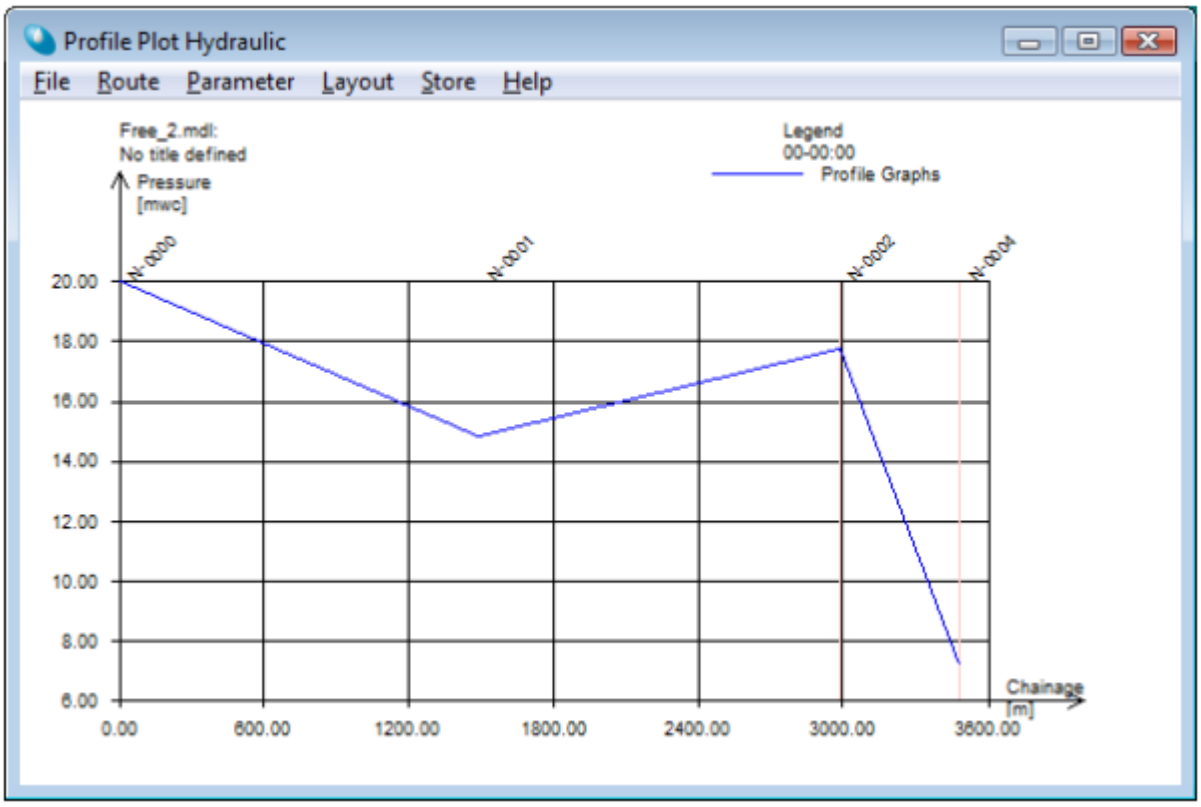


5. Click **OK** twice until you return to the Profile Plot Hydraulic dialog box.
6. Under the **Profile Plot Hydraulic** dialog box select the menu **Parameter** and then point to **Pressure**.

   The Parameter menu determines what to display in the plot. You can only display one route on a profile plot at a time.

   Under the File menu you can save the plot or export plots as a dxf file (AutoCad format). You can also open previously saved plots.

   View the result for the Pressure parameter.



Time Series plot

As mentioned previously this plot is not part of the scope for this lesson. It is, however, included in this Lesson 3 for Results since it represents the core area to monitor if your model includes dynamic behavior of daily demand profiles.

The exercise on the time series plot needs a model that contains more input data than the models you have created so far. The simulation requires for example a profile with demand data and to perform this exercise you will need to open one of the demo models supplied with the product.

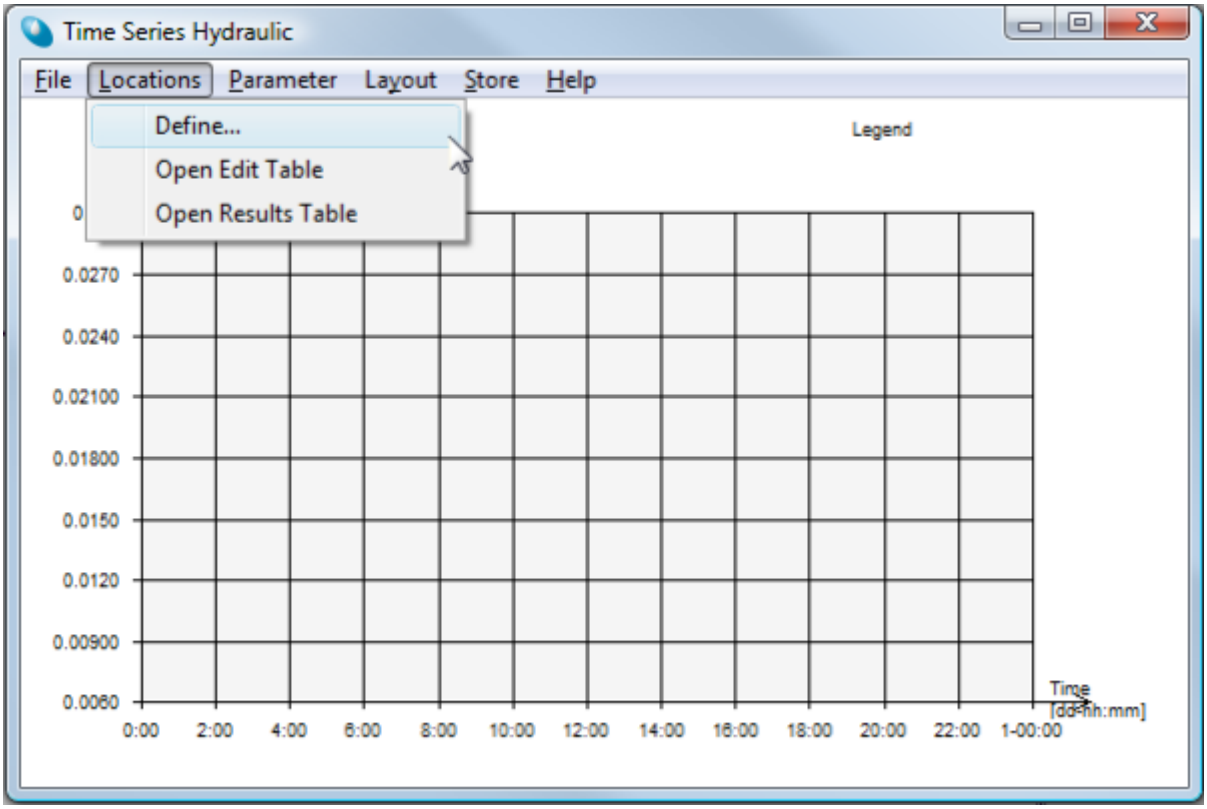
1. From the menu bar select **File** and then **Open** and locate the model called **demo.mdl**.

   If you made a default installation you will find the demo model here

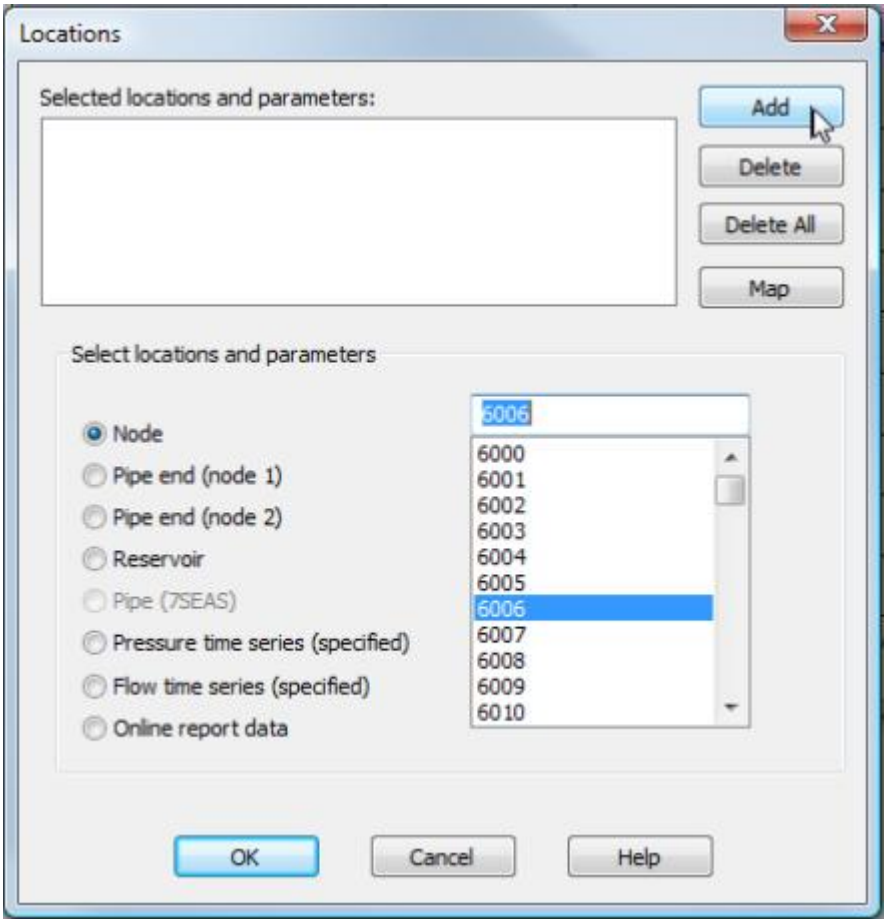
   C:\Program Files\7T\Aquis 1.5\Model
2. Perform a simulation, as explained in Lesson 2. Set the simulation period to perform a **one** (1) **day simulation** as shown in the following:

   Start time: **00-00:00**

   End time: **01-00:00**
3. Press **Run**.
4. Select **Results** and then **Time Series Plot**. This opens the Time Series Hydraulic dialog box.
5. Under the **Time Series Hydraulic** dialog box select **Locations** and point to **Define**.

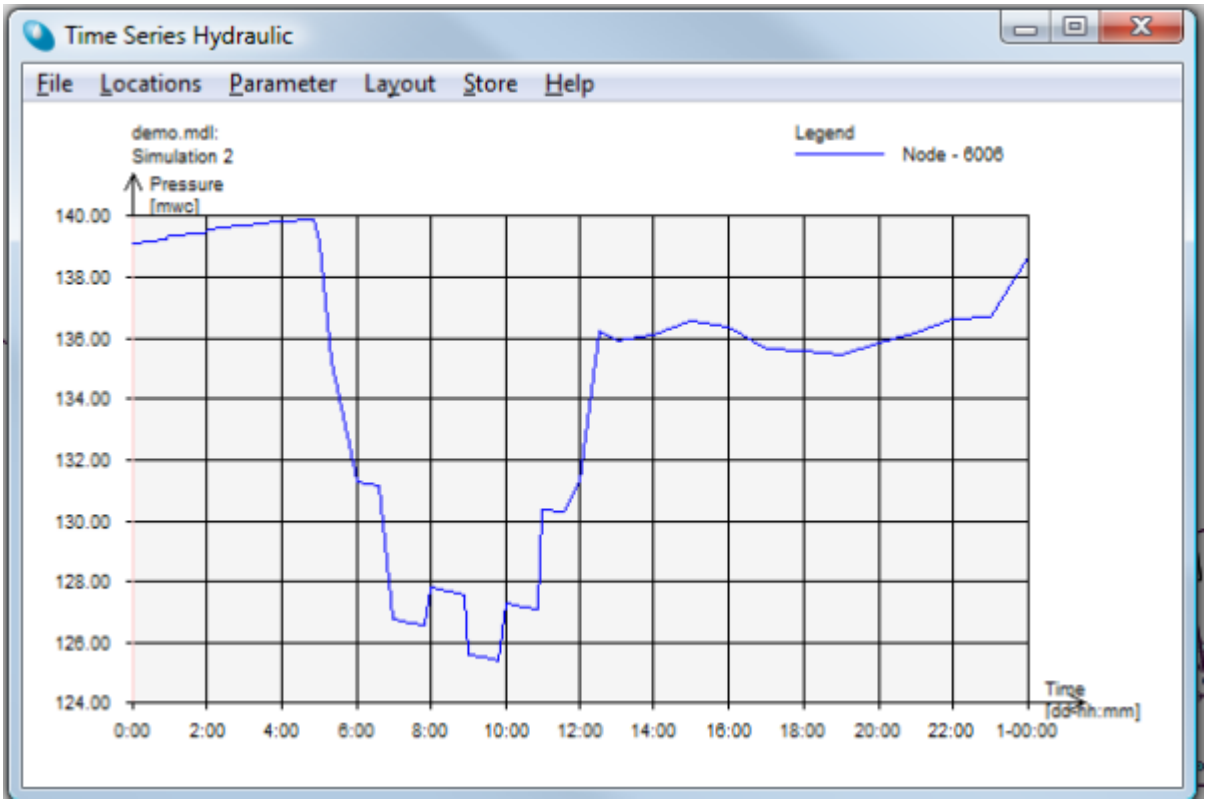


6. Under the **Locations** dialog box press **Delete All** to delete all existing locations.
7. Select the node **6006**.
8. Press **Add** and then **OK**.



9. Select **Parameter** and then **Pressure**.

   You will now see the following time series plot and how the node 6006 varies over time.



Option: View Files

The result files are available when the simulation is completed, but only if you have selected the appropriate checkboxes before the simulation. See Lesson 2 for details.

The simulation generates the following result files. There will be a short introduction to each report file. You are encouraged to peruse the files on your own.

- View Output File
- View Input File
- View Check File
- View CSP Check File

The result file displays in Windows Notepad. You can apply all standard Notepad functionality. The Find functions can be particularly useful if you are scanning through a rather long file.

On your computer you can locate the files under the following folder:

C:\Program Files\7T\Aquis 1.5\Calc

Output file

- From the menu bar select **Results** and then **View Files**, and point to **View Output File**.

This file contains in-depth information about all the model objects and their configuration.

In the Output file you can for example see the results for the model and the nodes.

S Y S T E M D A T A

= GENERAL DATA =

Actual directory : C:\Program Files\7T\Aquis 1.5\Calc\FREE_2

Name of input file: FREE_2

Number of pipes : 4
Number of nodes : 5

Pipes in pressurecontrol loops are marked %
Pipes in loops are marked *
Pipes to be dimensioned are marked #

= NODE RESULTS =

No.	Node Name	Flow l/s	Pressure mwc	Head mwc	Leakage flow l/s	Hydrant l/s	Supp. pressure mwc
1	N-0000	-3.50000	20.000	30.001	-	-	20.000
2	N-0001	0.50000	14.861	29.862	-	-	14.861
3	N-0002	1.50000	17.778	27.779	-	-	17.778
4	N-0003	0.50000	4.852	29.852	-	-	4.852
5	N-0004	1.00000	7.232	27.233	-	-	7.232

AQUIS 2009.08.04 14:31 Page: 9

Input file

- From the menu bar select **Results**and then **View Files**, and point to **View Input File**.

There are two file formats for the input file:

.DAT (contains all the pertinent and key input data for the model)

.sss (contains data that is only used for dynamic simulation - water behavior)

Example of a .DAT file

```
=========
= Title =
=========

*TITL
      "Simulation 2"
      " "
      " "
      " "
      " "
      /

*ASCI   Stop criteria : 1 1 1    /

======================
=   Valve Criteria   =
======================


*CMIN          Backflow Cv :      0.05000
    /

=================================================
=   Time Step Reservoir Level Change Criteria   =
=================================================


*WTDZ     Res. Pressure Change :      0.10000
    /

==============================
=   Min Time Step Criteria   =
==============================


*DTMI     Min. Time Step :           60
    /

======================
= Iteration Criteria =
======================


*ILIM          Basic Stop criteria :      0.00500
                  Basic Relaxation :      0.800
      ACV, Hydrant, etc Relaxation :      0.800
       ACV, Hydrant, Stop criteria :      0.00500
         Inlet pump characteristic :        1
```

Check file

- From the menu bar select **Results** and then **View Files** and point to **View Check File**.

The format of the Check file depends on the state of the model.

If there are no errors as a result of the compilation	...then the *.CHK file will show the following message at the end of the file: No errors during input!
If the error can only be reported as an Error Code	... then the error can only be reported as an Error Code, the *.CHK file will display something similar to this example. **** WARNING 2030 **** from LFLOW-3V module: ROUOUT Blind end in critical path!
If the error is caused by incorrect data definition	...then the *.CHK file will include the *.CF1 and the *.DAT file. When there is an error in these files, an error message will be written immediately after the incorrect data.

Example of a .CHK file

```
=========

= BATCH =

=========

*BATS     "C:\PROGRAM FILES\7T\AQUIS 1.5\CALC\FREE_2\FREE_2.SOP"       /

          >                                                    <        X



*STOP

 No errors during input!

Statistic:
Overall simulation time (sec):      0
Number of re-initializations:       1
Number of iterations:               8
```

POST-IT DATA LABELS

You can display specific node and pipe data for a visual presentation of the simulation results in the model. These are referred to as "post-it" data labels.

You can add various labels depending on your requirements. See the options in the table.

Post-it label	Description
Data labels	Displays results of the simulation.
Name labels	When you create a node or a pipe select the checkbox for **Out** to display the node name. *)
Comment labels	When you create a node or a pipe select the checkbox for **Out** to display the text. *)

*) These can be added without running a simulation.

This exercise only explains how to enable the display of data labels. You are encouraged to investigate the alternative options yourself.

Follow the guidelines in the ensuing exercise for Post-it labels to create your own data labels.

EXERCISE

This exercise provides you with an alternative approach to viewing results in your model. The exercise explains how to add data labels to nodes to provide you with an improved visual overview of the status in the operation of your network.

To complete this exercise you

- Have access to the demo file **Demo25.mdl**. If you made a default installation you will find the demo model here

  C:\Program Files\7T\Aquis 1.5\Model

To create a post-it data label

1. Open the model **Demo25.mdl**.
2. Run a simulation as explained in Lesson 2 and use the following definitions:

   Title: **Demo25**.

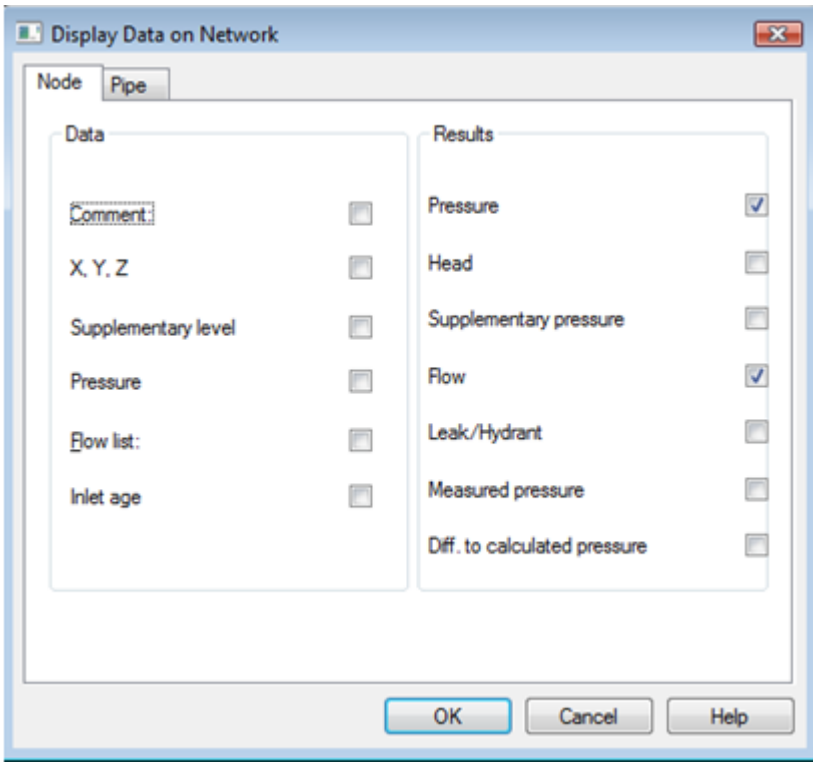
   Set the simulation period to **4 days**.

   Select all check boxes for result files .
3. From the menu bar select **View**>**Display Data Labels on Network**>**Select Data for All Labels**.

   This will activate the display of labels in your model.
4. Under **Display Data on Network** select the following checkboxes and click **OK**.

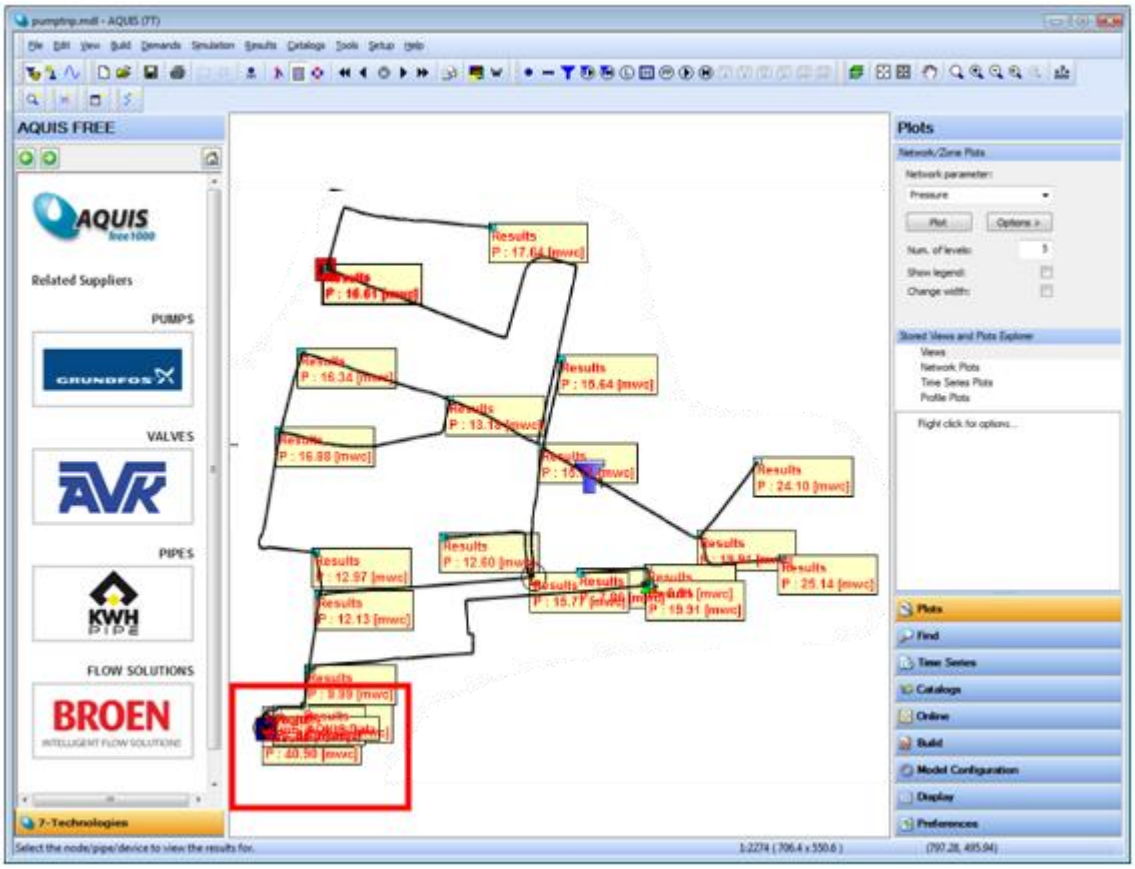
   Pressure

   Flow



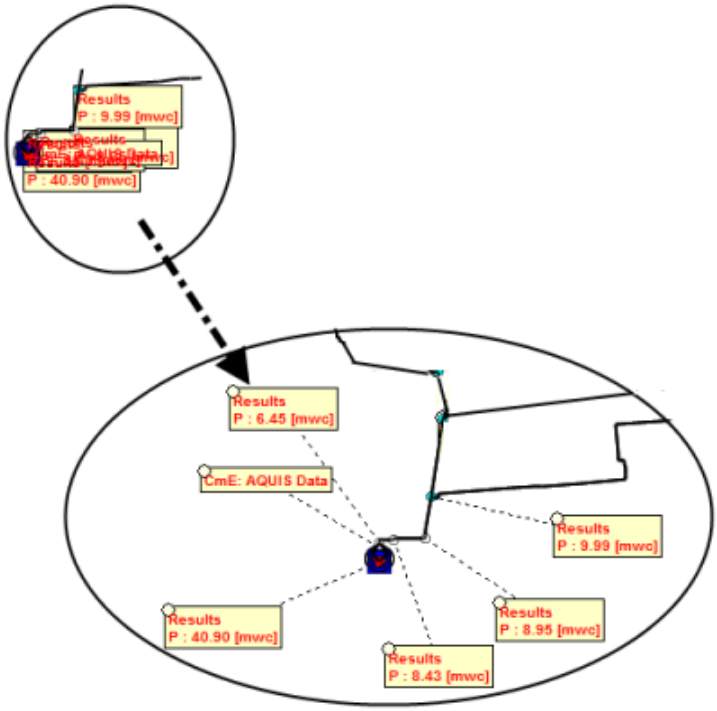
5. View the result.

As this demo model is a fairly large model the display of labels is far from pretty and looks very cluttered. Proceed to the next step to see how to improve the display of labels in the red frames.



6. From the menu bar select **Build**>**Move**.
7. Point to the circle in the upper left corner of the label and drag the label to the required position. This is illustrated in the following figure.

   Notice that you may inadvertently catch the focus of the node in which case you will be moving the node and not the label.



CONCLUSION

This lesson showed you how to view the results of the simulation.

You have now completed the basic introduction lessons to AQUIS free 1000.

You are recommended to investigate the possibilities of your own and use the online Help to assist you during the various operations.