# Design of Wastewater Collection System for Yatta Town Using Sewercad Software

**BY**

**BASHAR AL ASHHAB** **MOHAMMAD AL HALAWANI**

**CIVIL & ARCHITECTURAL ENGINEERING DEPARTMENT**

**COLLEGE OF ENGINEERING AND TECHNOLOGY**

**PALESTINE POLYTECHNIC UNIVERSITY**

**HEBRON- WEST BANK**

**PALESTINE**

**JANUARY 2008**

# Design of Wastewater Collection System for Yatta Town Using Sewercad Software

**BY**

**Bashar Al Ashhab** **Mohammad Al Halawani**

A PROJECT REPORT SUBMITTED IN PARTIAL FULFILLMENT OF THE
REQUIREMENTS FOR THE DEGREE OF
BACHELOR OF ENGINEERING
IN
CIVIL & ARCHITECTURAL ENGINEERING DEPARTMENT

SUPERVISED BY

**Dr. MAJED ABU SHARKH**

**CIVIL &ARCHITECTURAL ENGINEERING DEPARTMENT**
**COLLEGE OF ENGINEERING AND TECHNOLOGY**
**PALESTINE POLYTECHNIC UNIVERSITY**

**HEBRON- WEST BANK**
**PALESTINE**

**JANUARY 2008**

# CERTIFICATION

## Palestine Polytechnic University

(PPU)

**Hebron- Palestine**

The Senior Project Entitled:

# Design of Wastewater Collection System for Yatta Town Using Sewercad Software

**BY**

**BASHAR AL ASHHAB** **MOHAMMAD Al HALAWANI**

Prepared By:

*In accordance with the recommendations of the project supervisor, and the acceptance of all examining committee members, this project has been submitted to the Department of Civil and Architectural Engineering in the College of Engineering and Technology in partial fulfillment of the requirements of the department for the degree of Bachelor of Science in Engineering.*

**Project Supervisor** **Department Chairman**

**JANUARY 2008**

# إهـــــــــداء

إلى اقرب من في الوجود إلى نفسي ... والدي الحبيبين .

إلى أغلى من في الحياة على قلبي ... إخوتي الأعزاء .

إلى من أهدتني بهم السماء ... أصدقائي الأحباء .

إلى المنارات التي أضاءت لي الدرب ... أساتذتي الأجلاء .

إلى كل اللحظات السعيدة التي قضيناها داخل اسوار هذه الجامعة الغراء .

إلى أرواح كل الشهداء ... إلى فلسطين الإباء .

إلى كل شيء طاهر جميل في هذا الوطن المعطاء .

إلى كل هؤلاء ... أهدي ما جنيت بعناء .

بشار الأشهب

محـمـد الحـلـواني

# ACKNOWLEDGEMENT

We would like to express our thanks and gratitude to Allah, the Most Beneficent, the most Merciful who granted us the ability and willing to start and complete this project. We pray to his greatness to inspire us the right path to his content and to enable us to continue the work started in this project to the benefits of our country.

We wish to express our deep and sincere thanks and gratitude to Palestine Polytechnic University, the Department of Civil & Architectural Engineering, College of Engineering & Technology. We wish to express our thanks to Dr. Majed Abu-Sharkh, for a valuable help, encouragement, supervision and guidance in solving the problems that we faced from time to time during this project.

We can find no words to express our sincere, appreciation and gratitude to our parents, sisters and brothers, for their endless support and encouragement, we are deeply indebted to you and we hope that we may someday reciprocate it in someway.

**Work Team**

# ABSTRACT

# Design of Wastewater Collection System for Yatta Town Using Sewercad software

**BY**

**BASHAR AL ASHHAB** **MOHAMMAD HALAWANI**

***Palestine Polytechnic University***

There is no wastewater collection system at present inYatta town. The sewage from residential and public buildings in the area is drained to cesspits. These have become clogged with time and require frequent emptying. The continued use of cesspits with the icrease in population will cause environmental and health problem, and may create contamination of the underground water aquifer. Furthermore, emptying csspools constitutes an offensive odor nuisance to the population. On the other hand, emptying the vacuum trunks in the nearly wadi causes negative impacts on the visual landscape.

In reference to above description of the existing situation, there is a clear need of project in order to improve the sanitary level in Yatta town. The first step is serving the city with wastewater collection system. So, the main objective of this project is to design wastewater collection network for Yatta town of the Hebron district.

The present study considered the annual population growth and their water consumption for the coming 25 years that will be the design period, along with the commercial and industrial development in the area. The necessary hydraulic calculation needed for the design of the main trunks was carried out by simple calculation.

The results of the study show that wastewater disposal in Yatta area causes problems to the peoples; subsequently there is a big need for immediate steps for construction of the proposed wastewater collection system. Gravity flow sanitary sewer was proposed for most of Yatta areas to minimize the cost of construction and excavations with one pumping stations.

We used SewerCAd softwear in design the network of wastwater. SewerCAD is an extremely powerful program for the design and analysis of gravity flow and pressure flow through pipe networks and pumping stations. The program can be run in AutoCAD mode, giving you all the power of AutoCAD's capabilities, or in Stand-lone mode utilizing our own graphical interface. SewerCAD allows you to construct a graphical representation of a pipe network containing information such as pipe data, pump data, loading, and infiltration. You have a choice of conveyance elements including circular pipes, arches, boxes and more.

# TABLE OF CONTENTS


| Subject | Page |
|---|---|
| **TITLE** | i |
| **CERTIFICATION** | ii |
| **DEDICTION** | iii |
| **ACKNOWLEDGMENT** | iv |
| **ABSTRACT** | v |
| **TABLE OF CONTENTS** | vii |
| **LIST OF FIGURES** | ix |
| **LIST OF TABLES** | x |
| **CHAPTER ONE: INTRODUCTION** | 1 |
| 1.1 Background | 2 |
| 1.2 Problem Definition | 2 |
| 1.3 Purpose of Project | 3 |
| 1.4 Scope of the Work | 3 |
| 1.5 Project Area | 3 |
| 1.6 Stages of the Project | 6 |
| 1.7 Organization of the Report | 8 |
| **CHAPTER TWO: DESIGN AND PLANNING CRITERIA** | 9 |
| 2.1 Introduction | 10 |
| 2.2 Population Forecast | 10 |
| 2-2-1 Introduction | 10 |
| 2-2-2 Population projection | 11 |
| 2-2-3 Population Density | 12 |
| 2.3 Future Water Consumption | 12 |
| 2-3-1 Introduction | 12 |

2-3-2 Projected water consumption 13
2.4 Design Parameters 14
2-4-1 Flow rate projections 14
2-4-2 Hydraulic design 15
2-4-3 Minimum and maximum velocities 15
2-4-4 Pipes and sewers 16
2-4-5 Manholes and covers 16
2-4-6 Sewer slope 16
2-4-7 Depth of sewer pipe 16
2-4-8 Design period 17
2.5 Design and Planning Assumptions 17

**CHAPTER THREE: ANALYSIS AND DESIGN** 19
3.1 Introduction 20
3.2 Layout of the System 20
3.3 Design Computations 21
3-3 -1 Introduction 21
3-3-3 When to use SewerCAD 22
3.4 The Proposed Wastewater Collection System 23
3.5 Profiles of Sewer 23

**CHBTER FOUR : BILL OF QUANTITY** 24
Bill Of Quantity 25

**CHAPTER FIVE : CONCLUSION** 29
Conclusion 30

**REFRENCES** 32

# LIST OF FIGURES


| Figure No. | Description | Page |
| --- | --- | --- |
| 1-1 | Location Map of Yatta Town | 5 |

# LIST OF TABLES


| Table No. | Description | Page |
|---|---|---|
| 1-1 | Phases of the Project with their Expected Duration | 6 |
| 2-1 | Population forecast for Yatta town | 11 |
| 2-2 | Forecast Water Demand for Yatta  town | 13 |

**<u>CHAPTER</u>**

# *1* *INTRODUCTION*

---

*1.1 Background*

*1.2 Problem Definition*

*1.3 Purpose Of Project*

*1.4 Scope Of The Work*

*1.5 Project Area*

*1.6 Stages Of The Report*

*1.7 Organization Of The Report*

# CHAPTER ONE
# INTRODUCTION

## 1.1 Background

Currently, there are no public wastewater collection and treatment facilities serving most of rural areas in the West Bank. And due to the lack of sewage collection and treatment system, large areas in the West Bank and ground water aquifer are being contaminated by raw sewage. This contamination will have long-term impact on agricultural land and creates health hazards when utilized for human consumption. For the above mentioned reasons, serious and major steps should be taken to collect, dispose and treat the wastewater before discharging it in open environment.

## 1.2 Problem Definition

More than 60% of the water used for domestic purposes and industry turn into sewage requiring purification (treatment) for reuse in irrigation or alternative disposal. Contrarily, if wastewater not treated and not disposed of, sewage may contaminate sources of drinking water. In Palestine, water-borne diseases are very commonplace.

In the West Bank, no piped wastewater disposal system is available in most of the rural areas. Wastewater from individual residence is discharge directly into subsurface pits, allowing the wastewater to seep into the surrounding soil and percolate into the underlying aquifer causing ground water pollution. At the same time, the existing treatment plants are heavily overloaded and poorly operated and maintained.

Yatta town like other towns in West Bank has no sewage facility and the people are using latrines, cesspits and septic tanks for the disposal of wastewater. These latrines and cesspits are deteriorating and they are in very bad condition, adding to this the increasing in water consumption and consequently increasing in wastewater production, resulting in overflow from the cesspits and excessive recharges of ground water in Yatta area.

In view of this bad condition, and since there is no sewerage exist, along with the fast increase in the environmental and health problems, an evaluation and design of wastewater collection system study become a pressing necessity so as to solve all the problems that were mentioned above. This project which includes evaluation and design will consider the annual growth of the people and their water consumption for the coming 25 years, which will be the design period, along with the commercial and industrial development in the area.

## 1.3 Purpose of Project

The overall purpose of this project is to investigate and evaluate wastewater collection and treatment processes along with conceptual designs that are suitable for Yatta town. More specifically the main purposes of this project may be classified as follow:

1. Display the current situation of wastewater disposal in Yatta town.
2. Define the types of sewage facilities and their locations that will need to be constructed.
3. Propose wastewater collection system for the town and design the main trunks of the proposed sewerage collection network.
4. Estimate the cost for construction of the collection network.

The project will help in reducing the threat to the environment, water and land resources and to the health of the people living in Yatta town.

## 1.4 Scope of the Work

The scope of work of this project is to evaluate and develop preliminary conceptual design for sewer networks for Yatta town of the Hebron district. The preliminary design will incorporate a variety of design criteria including: investigation of site, site suitability, design alternatives, environmental consideration and cost estimate.

## 1.5 Project Area:

Yatta town is located 12km south of the Hebron city as shown on the project location map Figure 1.1. The total population within the administrative borders in year 2005 is about 62000. The elevation within the area ranges from 850-750 m with respect to sea level.

The average annual rainfall is around 300 mm and the minimum average annual temperature rises to 7.3 ºC and the maximum average annual temperature rises to 22.9 ºC. The per capita water consumption for domestic use does not exceed 80 liter per day. The present total administrative area of the town is about 24552.76 dounm, where building for population occupies 32 $Km^2$.

## 1.6 Stages of the Project

The project consists of six phases, which are proposed to be completed in accordance with time schedule shown in (Table 1.1). The description of each of the six phases of the project and tasks involved is listed below:

**Table (1.1): Phases of the Project with their Expected Duration**

| Phase No. | Title | Duration | | | | | | | | |
|---|---|---|---|---|---|---|---|---|---|---|
| | | 2/07 | 3/07 | 4/07 | 5/07 | 6/07 | 9/07 | 10/07 | 11/07 | 12/07 |
| 1 | Collection and Analysis of Data | ■ | ■ | ■ | | | | | | |
| 2 | Surveying Works. | | | ■ | ■ | | | | | |
| 3 | Design of the Sewage Network. | | | | ■ | ■ | ■ | | | |
| 4 | Draw layouts and Profiles. | | | | ■ | ■ | ■ | | | |
| 5 | Preparing Bill of Quantities | | | | | | ■ | ■ | | |
| 6 | Writing the Report. | | | | | | ■ | ■ | ■ | ■ |

**Phase 1: Collection and Analysis of Data**

During this phase, available data and information were collected from different sources. Moreover, many site visits to the project area were undertaken. First phase included the following tasks:

1. Collection of aerial and topographical maps of the area.
2. Collection, analysis and augmentation as necessary data on population, land use, zoning, water consumption and environmental conditions.

**Phase 2: Perform the Surveying Works**

The tasks which were performed in the second phase are:

1. Determine the coordinates of points using Geographical Information System.
2. Evaluate of the contour maps and matching it with actual ground levels.
3. Performing and selecting topographic survey for the sewage network.

**Phase 3: Design of the Sewage Network**

During the third phase, the areas to be served by sewage were defined, the layout was established, and the necessary hydraulic calculations needed for the design of one of the main trunks were carried out. The tasks, which were performed in this phase, are:

1. Define the area to be served by sewerage and establish the boundaries.
2. Establish a system layout which includes the areas that are going to be served, existing streets and roads, topography ….. etc.
3. Establish the main catchments areas and routes of the sewer.
4. Prepare a design criterion that meets the sewage contribution and flow for the entire area through the year 2032.
5. Do the necessary hydraulic calculation and find out the sewers diameter.

**Phase 4: Preparing Plan Drawings and Profiles**

Plan drawings and profiles with appropriate scales for the wastewater collection system were prepared.

**Phase 5: Preparing Bill of Quantities and Cost Estimates**

After finishing the design calculation of the main trunks, the research team prepared bill of quantities and estimate the cost of the project.

**Phase 6: Writing the Report**

Upon the completion of the work, one final report was written and submitted to the Department of Civil and Architectural Engineering at Palestine Polytechnic University.

## 1.7 Organization of the Report

The study report has been prepared in accordance with the objectives and scope of work. The report consists of five chapters.

The first chapter entitled “Introduction” outlines the problem, purpose of project, scope of the work and phases of the project.

Chapter two entitled “Design and Planning Criteria” presents information about population and their densities, the actual water consumption, land use, and design criteria applicable to the sewerage networks.

Chapter three entitled “Analysis and Design” Design the Wastewater system by using SewerCAD software and preparing the profiles of sewer lines .

Chapter Four entitled "Bill of Quantity" After finishing the design the team prepared bill of quantity.

Chapter five , which is the last chapter, entitled “Conclusions” summarized the project into briefly notes.

**CHAPTER**

# 2 *DESIGN AND PLANNING CRITERIA*

---

***2.1 Introduction***

***2.2 Population Forecast***

***2.3 Future Water Consumption***

***2.4 Design Parameters***

***2.5 Desing And Planning Assumption***

# CHAPTER TWO
# DESIGN AND PLANNING CRITERIA

## 2.1 Introduction

In the previous chapters, the problem of the study area has been defined and the objectives of the project have been listed. The characteristics of the project area (Yatta town) have been described. In this chapter, basis for planning and design will be discussed including present population, population forecasting, projected water consumption, town structure plan, and the design and planning criteria of the project.

## 2.2 Population Forecast

### 2.2.1 Introduction

The ideal approach for population forecasting is by the study and use of previous census records, which cover along period. The longer the period, and the more comprehensive the census data, the more accurate will be the results, which will be obtained. In the analysis of these data, demographical, economical and political factors should be considered in order to develop a method of forecasting which will predict the expected growth rate, future population and its distribution in the different zones of the area under consideration.

In Yatta town, there is great uncertainty in the political and economical future. Additionally, there were no accurate population data since the occupation of the West Bank in 1967, until 1997 when the Palestinian Central Bureau of Statistic (PCBS) conducted comprehensive census covering the West Bank and Gaza Strip. The final results of this census show that the total population of Yatta town is 50000 inhabitants.

Due to the unstable condition of the area during the last 50 years, it would be very difficult to develop a statistical interpretation to extrapolate future population. Some reasonable assumptions have, therefore, been made to project the future population of Yatta town over the next 25 years.

### 2.2.2 Population projection

Prediction of the future population of Yatta town is very difficult due to the lack of reliable historic data, and the political uncertainties, which will greatly influence future social and economic development. At the same time, the available data on past population growth do not constitute a reliable basis for projecting the future population growth in Yatta town.

The base for the forecast is the 1997 population for Yatta town obtained from PCBS of 50000 inhabitants. The annual growth rates for the next twenty years are also obtained from the PCPS and they are presented in Table 2.1.

To calculate the population at the end of the design period (year 2032), a geometric increase is assumed, represented by the following equation:

$$P = P_0 * (1+ r)^{n} \qquad (2.1)$$

Where, P is the future population, $P_0$ is the present population, r is the annual population growth rate, and n is the period of projection.

Using the above assumption and equation, Table 2.1 presents the population projection up to the design horizon of 2032. The data show that the population of Yatta town is estimated to be 90568 in year 2032.

**Table (2.1) : Population Forecast for Yatta Town.**

| Year | Annual Growth Rate(%) | Population |
|---|---|---|
| **2000** | **4.0** | **56244** |
| **2005** | **3.5** | **65841** |
| **2010** | **3.0** | **73427** |
| **2015** | **2.5** | **77983** |
| **2020** | **2.25** | **83412** |
| **2032** | **2.00** | **90568** |

### 2.2.3 Population Density

When determining the density of population, it is either related to the total municipal area (gross density) or to the built–up area only (net density). The gross density related to the municipal area includes large industrial areas, agricultural areas, un-built areas, public parks, large water surfaces, forests …etc. The net density is related to the built up urban area, but it includes small-scale industries, schools, public and commercial buildings, and roads.

Sewer design, however, is based on the net densities of population, because the provision of sewers is limited to the built-up areas. The net density of population varies considerably from district to district. Population densities are based on the town structure plan , which serves for issuing building permit There are no studies done concerning the population densities in Yatta town. The population density in Yatta town is nearly 250 capita / hectare.

## 2.3 Future Water Consumption

### 2.3.1 Introduction

Water consumption is not constant, yearly, monthly, weekly, daily and hourly variations in water consummations are observed. Certain dry years cause more consumption. In hot months water is consumed in drinking, bathing, and watering lawns and gardens. On holidays and weekends the water consumption may be high. Even during day water use various with high use during morning hours and close to noon and low use at night. Maximum daily demand or maximum daily consumption usually occurs during summer months. The ideal approach to assess the existing and future per capita water consumption is by analyzing and extrapolating the available record on water consumption and demand in conjunction with the expected social and economical development. This approach can be adopted in areas having continuous supply systems where reliable information about population, population distribution and demand are known. There are problems adopting this approach for Hebron area including Yatta town due to insufficient data and also the intermittent water supply.

Restrictions on the Palestinian use of the annual ground water resources of the West Bank led to limited quantities availability of water and due to this condition; the average consumption of water in Yatta town for all purposes does not exceed 25 cubic meters per capita per year. Given these circumstances, the approach to determine per capita water consumption depends on the analysis of the existing information. The existing per capita consumption has already been assessed at (110 liter/capita.day).

### 2.3.2 Projected water consumption

The present average consumption of water for domestic use in Yatta town does not represent the present and actual demand of water. So, it is estimated the water consumption in Yatta will dramatically increase during the next few years, due to several ongoing water projects, which are:

1. Drilling new production wells;
2. Upgrading the existing water supply system;
3. Rehabilitation of the existing network; and
4. Detecting network leakage as well as dealing with institutional development of water sector and tariff structures.

The Yatta municipality has estimated the existing per capita water consumption at about 100 l/c.d for year 2005, and the rate of increase in the annual water consumption per capita to 1.5 %. It is estimated that the per capita water supply will be 150 l/c.d including the physical losses in the year 2032. It should be noted that the figure includes commercial and industrial consumption. The estimated per capita water consumptions for the year 2005, 2010, 2015, 2020 and 2032, is presented in Table 2.2.

**Table (2.2): Forecast Water Demand for Yatta Town.**

| Year | Population | Water Demand ($m^3$/year) | | Water Demand(l/c.d) |
|---|---|---|---|---|
| | | Per Capita | Total | |
| 2005 | 65841 | 36.5 | 2,403,197 | 100 |
| 2010 | 73427 | 39.3 | 2,885,681 | 107.7 |
| 2015 | 77983 | 42.34 | 3,301,800 | 116 |
| 2020 | 83412 | 45.63 | 3,806,090 | 125 |
| 2032 | 90568 | 54.75 | 4,958,598 | 150 |

It may be noted from Table 2.2 that the projected water consumption for the design period (year 2032) is 150 liter per capita per day, in which 80% is returned to sewer (120 l/c.d.).

## 2.4 Design Parameters

### 2.4.1 Flow rate projections

The total wastewater flow in sanitary sewers is made up of two main components: (1) Residential. (2) Infiltration. Sanitary sewers are designed for peak flows from residential and peak infiltration allowance for the entire service area. The flow rate projections are necessary to determine the required capacities of sanitary sewers. These projections will be based on:

1. Population: Future population at the end of design period should be estimated. The estimated population of Yatta town in the year 2032 is 90568 inhabitants.
2. The present domestic water consumption and future consumption.
3. The percentage of water going to the sewer: In general, the average wastewater flow may vary from 60 to 90 percent of the water used in the community. A value of 80 percent has generally been agreed upon by all the authors of earlier projects in the West Bank and other locations under similar conditions.
4. The service connection percentage: The percentage of houses that will be served by sewers will depend on the nature of the habitat in the catchments area considered and of the design period. It has been assumed that the service connections will increase, to full coverage for the urban population in year 2032
5. The uncontrolled inflow and infiltration: Infiltration is the entrance to the collection system of water from outside sources such as groundwater. Inflow is the entrance to the collection system of runoff during a rainfall event. Infiltration depends mainly on the state of the network; the depth at which it is buried and the groundwater elevations. Most of the sewers to be laid will be new and the ground water elevations in the area are low. Ground water infiltration seems then to be not significant. The network will be designed to avoid rainwater inflow. However, there will always be cases of manhole leaks, loose joints and private individuals who link up their rainwater pipes to the sewerage network. Given the difficulty of accurately estimating these parameters

and according to previous studies and data of another area under similar conditions, a mean discharge increase of liter per second per hectare will be applied when dimensioning the sewerage system.

6. The peak coefficient : In general, this coefficient increases when the rate of connected population decrease, for example when the flow rate is weak. In the other hand, when the connected population is important, the variation around a mean discharge is weaker. As there are few field investigations conducted in the study area to estimate this factor; it will be determined from the practice and experience of the designer. The following relation has been used commonly by the designer and gives satisfactory results:

$$Pf = 1.5 + 2.5 / \sqrt{q} \tag{2.1}$$

Where, q (in l/s) is the daily average flow rate of the network branch under consideration and Pf is the peak factor.

Using these assumptions, the flow rate projections were evaluated for the study area (Yatta town).

### 2.4.2 Hydraulic design

As mentioned earlier and according to usual practice, the sewers will be designed for gravity flow using Manning's formula:

$$V = 1/n \, R^{2/3} \, S^{1/2} \tag{2.2}$$

Depending on pipe materials, the typical values of n are:

- Reinforced Concrete (RC) n = 0.013
- Polyvinyl Chloride (PVC) n = 0.011
- Ductile Iron: n = 0.013
- Asbestos Cement: n = 0.012

### 2.4.3 Minimum and maximum velocities

For a circular sewer pipe, the velocity at half-depth is equal to the velocity at full-depth. To prevent the settlement of solid matter in the sewer, the literature suggested that the minimum velocity at half or full depth – during the peak flow period – should not be less than 0.6 m/s, but point out that the minimum self cleansing velocity of 1.0 m/s is to be preferred wherever this is practicable. Usually, a maximum sewer velocity are limited to about 3 m/s in order to limit abrasion and avoids damages which may occur to the sewers and manholes due to high velocities.

### 2.4.4 Pipes and sewers

i) Necessary because some large objects such as scrub brush, sometimes gets into sewers. Experience indicates a minimum diameter of 200 mm (8 in) for sewer pipes. For house connections, smaller sizes may be used.

ii) Pipe Materials: different pipe materials may be recommended for the sewers.

1. Polyvinyl chloride PVC, vitrified clay VCP or polyethylene PE material for small size pipes (approximately up to the size 400 mm in diameter).
2. Centrifugal cast reinforced concrete pipes may be used for larger diameter.

### 2.4.5 Manholes and covers

Manholes should be located at changes in size, slope direction or junction with secondary sewer. Manholes spacing generally does not exceed 60 m and should never be greater than 100 m except in sewers which can be walker through gravity. The minimum cover over sewer line will be of 1.5 m, for the buried section.

### 2.4.6 Sewer slope

For a circular sewer pipe, the slope must be between the minimum and maximum slope, the minimum and maximum slope is determined from minimum and maximum velocity. Generally the natural ground slope is used because it is the technical and economic solution, the solution is therefore recommended.

### 24.7 Depth of sewer pipe

As mentioned earlier, the depth of sewers is generally 1-2 m below the ground surface. Depth should be enough to receive the sewage by gravity, avoid excessive traffic loads, and avoid the freezing of the sewer. It is recommended that the top of sewer should not be less than 1 m below basement floor.

### 2.4.8 Design period

Sewers are designed on estimated future flows at the end of a design period. So the design period is thus the length of time throughout which the capacity of a sewer will be able to cope with the expected flows and may be assumed at:

1. Drains (concrete): 20 – 30 years.
2. Sanitary sewers: 25 – 30 years.
3. Pumping station: Equipment: 15years.
4. Buildings: 25 – 30 years.

The design period adopted for this project is 25 years.

## 2.5 Design and Planning Assumptions

The design and planning assumptions used in this project are as follow:

1. Design period 25 year (from 2007-2032).
2. Present (2005) population of municipality of Yatta town is 65841capita.
3. The growth rate will be 2%-4.5%.
4. The existing per capita water consumption has been assessed 80 l/c.d.
5. Total administrative area of municipality of Yatta town 24552.76 dounm.
6. Future 2032 population of Yatta town 90568 capita.
7. Per capita water consumption by 2032 will reaches 150 l/c.d.
8. The wastewater production is about 80% of their water consumption.
9. Formula to be used in design of sewers :(Manning formula)

$$V=(1/n)^* R^{2/3} {}^*S^{1/2} \qquad (2.2)$$

10. Minimum velocity 0.6 m/sec.
11. Maximum velocity 3.0 m/sec.
12. h/D = 0.5 for main trunks.
13. Maximum manhole spacing 60 m for main trunk.
14. Minimum pipe diameter = 8 inch (200 mm)

15. Infiltration rate 20 % of average domestic wastewater.
16. Peak factor determine by equation

$$P_f = 1.5 + (2.5 / \sqrt{q}) \quad (2.1)$$

17. Depth of sewer pipe: Minimum covers not less than 1.5 from the crown.
18. Maximum slope Smax = 15 %
19. Minimum slope Smin = 0.5%

**CHAPTER**

# *3* *ANALYSIS AND DESIGN*

---

*3.1 Introduction*

*3.2 Layout Of The System*

*3.3 Design Computations*

*3.4 The Proposed Wastewater Collection System*

*3.5 Profiles of Sewer*

# CHAPTER THREE
# ANALYSIS AND DESIGN

## 3.1 Introduction

In this project, an attempt is made to evaluate and design wastewater collection system for Yatta town, and develop a future plans for construction of the collection system, corresponding to population growth and the water consumption and subsequently the wastewater production from different sources in the future, in order to reduce the problem causes by the disposal of raw wastewater in the area. In this chapter, the layout of the system established is presented, and the computation procedures and tables are given along the drawings of layout and profiles for all the lines designed.

## 3.2 Layout of the System

The first step in designing a sewerage system is to establish an overall system layout that includes a plan of the area to be sewered, showing roads, streets, buildings, other utilities, topography, soil type, and the cellar or lowest floor elevation or all buildings to be drained. Where part of the drainage area to be served is undeveloped and proposed development plans are not yet available , care must be taken to provide adequate terminal manholes that can later be connected to the system constructed serving the area .

In establishing the layout of wastewater collection system for Yatta area, the following basic steps were followed:

1. Obtain a topographic map of the area to be served.
2. Locate the drainage outlet. This is usually near the lowest point in the area and is often along a stream or drainage way.
3. Sketch in preliminary pipe system to serve all the contributors.
4. Pipes are located so that all the users or future users can readily tap on. They are also located so as to provide access for maintenance and thus are ordinarily placed in streets or other rights-of-way.

5. Sewers layout is followed natural drainage ways so as to minimize excavation and pumping requirements. Large trunk sewers are located in low-lying areas closely paralleling streams or channels.
6. Establish preliminary pipe sizes. Eight inches pipe size (usually the minimum allowable) can serve several hundred residences even at minimal grades.
7. Revise the layout so as to optimize flow-carrying capacity at minimum cost. Pipe lengths and sizes are kept as small as possible, pipe slopes are minimized, and followed the ground surface slope to minimize the depth of excavation, and the numbers of appurtenances are kept as small as possible.
8. The pumping is avoided across drainage boundaries. Pumping stations are costly and add maintenance problems.

**The final layout of the wastewater collection system for Yatta town is given in attached drawings.**

## 3.3 Design Computations

### 3.3.1 Introduction

The detailed design of sanitary sewers involves the selection of appropriate pipe sizes and slopes to transport the quantity of wastewater expected from the surroundings and upstream areas the next pipe in series, subject to the appropriate design constrains. The design calculations necessary for Yatta sanitary sewers are performed using SewerCAD Vs5 software. This computer program is develop by the Haestad Methods, Inc. More detailed about this program is given below.

### 3.3.2 What is SewerCAD?

SewerCAD is an extremely powerful program for the design and analysis of gravity flow and pressure flow through pipe networks and pumping stations. The program can be run in AutoCAD mode, giving you all the power of AutoCAD's capabilities, or in Stand-lone mode utilizing our own graphical interface. SewerCAD allows you to construct a graphical representation of a pipe network containing information such as pipe data, pump data, loading, and infiltration. You have a choice of conveyance elements including circular pipes, arches, boxes and more.

The gravity network is calculated using the built-in numerical model, which utilizes both the direct step and standard step gradually varied flow methods. Flow calculations are valid for both surcharged and varied flow situations, including hydraulic jumps, backwater, and drawdown curves. You also have the flexibility to mix gravity and pressure components freely, building your systems in parallel or in series as they exist in the field. Pressure elements can be controlled based on system hydraulics, turning pumps on and off due to changes in flows and pressures.

SewerCAD's flexible reporting feature allows you to customize and print the model results in both a report format and as a graphical plot.

### 3.3.3 When to Use SewerCAD?

SewerCAD is so flexible you can use it for all phases of your project, from the feasibility report to the final design drawings and analysis of existing networks. During the feasibility phase, you can use SewerCAD to create several different system layouts with an AutoCAD or Micro Station drawing as the background, or within AutoCAD itself. For the final design, you can complete detailed drawings with notes that can be used to develop the construction plans. In summary, you can use SewerCAD to:

- Design multiple sanitary sewer systems.
- Analyze various design scenarios for sanitary sewer systems.
- Import and export AutoCAD and Micro Station .DXF files.
- Generate professional-looking reports for clients.
- Generate plan and profile plots of a network.

**The computation tables for the design of the proposed wastewater collection system are given in attached report**.

## 3.4 The Proposed Wastewater Collection System

In the proposed study for the wastewater collection system for Yatta town, the trial is made to design the main trunks of the collection system for year 2032. There are nine main trunks. This section deals with the results of the suggested wastewater collection network for year 2032.

The appropriate pipe diameters, lengths, land slopes, and location of the manholes are found by doing the calculations given in the previous section. During and once the sewer design computations have been completed, alternative alignments have be examined, and the most cost–and energy–effective alignment has been selected. The final results for the appropriate diameters for the proposed wastewater collection system, slopes and lengths of the pipes are given in attached tables along with all relevant data. The calculated velocities, flow rates, and depth of flow in pipes are given in the same tables..

It is observed form the tables and figures that the collection network covers most of the area of Yatta, the slope of the pipes follow in most cases the slope of the ground.

## 3.5 Profiles of Sewer

The profiles of sewer area assist in the design and are used as the basis of construction drawings. The profiles are usually prepared for each sewer line at a horizontal and vertical scale. The profile shows the ground or street surface, tentative manhole locations, elevation of important subsurface strata such as rock, locations of borings, all underground structures, basement elevations, and cross streets. A plan of the line and relevant other structures are usually shown on the same street (McGhee, 1991).

After all the calculation is completed and all the maps of the proposed collection system are prepared, detailed profiles for each sewer is drawn. The profiles of sewer lines are shown in attached figures. These profiles had shown the ground elevation, the proposed sewer lines, manholes (manholes number and the spacing between the manholes), depth of excavations, the diameters and slopes of the pipes, and the type of soil.

CHAPTER

*4* BILL OF QUANTITY

## BILL OF QUANTITY

# CHAPTER FIVE

# BILL OF QUANTITY
# FOR THE PROPOSED WASTEWATER COLLECTION SYSTEM

## 1 – Excavations and backfilling

| Item | Item Description | Unit | Quantity | Unit Price € | Total amount € |
|---|---|---|---|---|---|
| **1.1** | Excavations and backfilling nominal pipe diameter **1050** mm | | | | |
| 1.1.1 | Excavation of pipe trench in all kinds of soil, rock, etc, the price shall include cost of backfilling with selected suitable material approved by the engineer, and shall include leveling and compaction to a depth not exceeding 2 m and disposing surplus material outside the site according to drawings. And specifications. | L.m | **648. 5** | | |
| 1.1.2 | Ditto, but for excavations between 2.00-2.50m | L.m | **138** | | |
| 1.1.3 | Ditto, but for excavations between 2.50-4m | L.m | **2067** | | |
| **1.2** | Excavations and backfilling nominal pipe diameter **900** mm | | | | |
| 1.2.1 | Excavation of pipe trench in all kinds of soil, rock, etc, the price shall include cost of backfilling with selected suitable material approved by the engineer, and shall include leveling and compaction to a depth not exceeding 2 m and disposing surplus material outside the site according to drawings. And specifications. | L.m | **1431** | | |
| 1.2.2 | Ditto, but for excavations between 2.00-2.50m | L.m | **408.5** | | |
| 1.2.3 | Ditto, but for excavations between 2.50-4.00m | L.m | **161** | | |
| **1.3** | Excavations and backfilling nominal pipe diameter **750 mm** | | | | |
| 1.3.1 | Excavation of pipe trench in all kinds of soil, rock, etc, the price shall include cost of backfilling with selected suitable material approved by the engineer, and shall include leveling and compaction to a depth not exceeding 2 m and disposing surplus material outside the site according to drawings. And specifications. | L.m | **1465** | | |
| 1.3.2 | Ditto, but for excavations between 2.00-2.50m | L.m | **347** | | |
| 1.3.3 | Ditto, but for excavations between 2.50-4.00m | L.m | **305** | | |
| **1.4** | Excavations and backfilling nominal pipe diameter **600 mm** | | | | |
| 1.4.1 | Excavation of pipe trench in all kinds of soil, rock, etc, the price shall include cost of backfilling with selected suitable material approved by the engineer, and shall include leveling and compaction to a depth not exceeding 1.5m and disposing surplus material outside the site according to drawings. And specifications. | L.m | **1540** | | |
| 1.4.2 | Ditto, but for excavations between 2.00-2.50m | L.m | **545.5** | | |
| 1.4.3 | Ditto, but for excavations between 2.50-4 m | L.m | **1075** | | |

| Item | Item Description | Unit | Quantity | Unit Price € | Total amount € |
|---|---|---|---|---|---|
| **1.5** | Excavations and backfilling nominal pipe diameter **450** mm | | | | |
| 1.5.1 | Excavation of pipe trench in all kinds of soil, rock, etc, the price shall include cost of backfilling with selected suitable material approved by the engineer, and shall include leveling and compaction to a depth not exceeding 2 m and disposing surplus material outside the site according to drawings. And specifications. | L.m | **1791** | | |
| 1.5.2 | Ditto, but for excavations between 2.00-2.50m | L.m | | | |
| 1. 5.3 | Ditto, but for excavations between 2.50-4.00m | L.m | | | |
| **1.6** | Excavations and backfilling nominal pipe diameter **375** mm | | | | |
| 1.6.1 | Excavation of pipe trench in all kinds of soil, rock, etc, the price shall include cost of backfilling with selected suitable material approved by the engineer, and shall include leveling and compaction to a depth not exceeding 2 m and disposing surplus material outside the site according to drawings. And specifications. | L.m | **2065** | | |
| 1.6.2 | Ditto, but for excavations between 2.00-2.50m | L.m | **255** | | |
| 1.6.3 | Ditto, but for excavations between 2.50-4.00m | L.m | **32.5** | | |
| **1.7** | Excavations and backfilling nominal pipe diameter **300 mm** | | | | |
| 1.7.1 | Excavation of pipe trench in all kinds of soil, rock, etc, the price shall include cost of backfilling with selected suitable material approved by the engineer, and shall include leveling and compaction to a depth not exceeding 2 m and disposing surplus material outside the site according to drawings. And specifications. | L.m | **1810** | | |
| 1.7.2 | Ditto, but for excavations between 2.00-2.50m | L.m | **28** | | |
| 1.7.3 | Ditto, but for excavations between 2.50-4.00m | L.m | | | |
| **1.8** | Excavations and backfilling nominal pipe diameter **250 mm** | | | | |
| 1.8.1 | Excavation of pipe trench in all kinds of soil, rock, etc, the price shall include cost of backfilling with selected suitable material approved by the engineer, and shall include leveling and compaction to a depth not exceeding 2 m and disposing surplus material outside the site according to drawings. And specifications. | L.m | **1857** | | |
| 1.8.2 | Ditto, but for excavations between 2.00-2.50m | L.m | | | |
| 1.8.3 | Ditto, but for excavations between 2.50-4.00m | L.m | | | |
| **1.9** | Excavations and backfilling nominal pipe diameter **200 mm** | | | | |
| | Excavation of pipe trench in all kinds of soil, rock, etc, the price shall include cost of backfilling with selected suitable material approved by the engineer, and shall include leveling and compaction to a depth not exceeding 2 m and disposing surplus material outside the site according to drawings. And specifications. | Lm | **6730** | | |
| 1.9.1 | Ditto, but for excavations between 2.00-2.50m | Lm | **198. 5** | | |
| 1.9.2 | Ditto, but for excavations between 2.50-4.00m | Lm | **251. 5** | | |

## 2– Pipes

| Item | Item Description | Unit | Quantity | Unit Price € | Total amount € |
|---|---|---|---|---|---|
| | **Pipes** | | | | |
| **2.1** | Supply, store and installation of pipes diameter **1050(uPVC)**with the Techen stamp or equivelant, along with the fittings, according to drawings, and specifications. | L.m | **2851** | | |
| 2.2 | Ditto, but for pipes diameter **900mm(u PVC)** | L.m | **2001** | | |
| 2.3 | Ditto, but for pipes diameter **750mm(u PVC)** | L.m | **2171** | | |
| 2.4 | Ditto, but for pipes diameter **600mm(u PVC)** | L.m | **3150** | | |
| 2.5 | Ditto, but for pipes diameter **450mm(u PVC)** | L.m | **1791** | | |
| 2.6 | Ditto, but for pipes diameter **375mm(u PVC)** | L.m | **2504** | | |
| 2.7 | Ditto, but for pipes diameter **300mm(u PVC)** | L.m | **1867** | | |
| 2.8 | Ditto, but for pipes diameter **250mm(u PVC)** | L.m | **1884** | | |
| 2.9 | Ditto, but for pipes diameter **200mm(u PVC)** | L.m | **6772** | | |

## 3 – Concrete manholes

| Item | Item Description | Unit | Quantity | Unit Price € | Total amount € |
|---|---|---|---|---|---|
| **3** | **Concrete manholes**<br>**Precast manholes** | | | | |
| 3.1 | Supplying and installation of manhole, coated with coal tar epoxy, including excavations in all kinds of soil, rock, etc, shall include cost of backfilling with selected suitable material approved by the engineer, and steps and benching, heavy duty cover 25 tons for streets, and 8 tons for cross country fields and backfilling not exceeding 2m .**Diameter1200mm** according to drawings and specifications. | Nr. | **107** | | |
| 3.2 | Ditto, but depth between 2.00-2.50m | Nr. | **393** | | |
| 3.3 | Ditto, but depth between 2.50-4.00m | Nr. | **262** | | |

## 4 – Pipe bedding

| Item | Item Description | Unit | Quantity | Unit Price € | Total amount € |
|---|---|---|---|---|---|
| **4** | **Pipe bedding** | | | | |
| 4.1 | Supplying, installation and compaction of (**Absolet**) fine granular material, under, above and around pipe Diameter (according to depth at items 1.1) **1050mm** according to the drawings and specifications. | L.m | **2851** | | |
| 4.2 | Ditto, but for pipes diameter 900mm | L.m | **2001** | | |
| 4.3 | Ditto, but for pipes diameter 750mm | L.m | **2171** | | |
| 4.5 | Ditto, but for pipes diameter 600mm | L.m | **3150** | | |
| 4.6 | Ditto, but for pipes diameter 450mm | L.m | **1791** | | |
| 4.7 | Ditto, but for pipes diameter 375mm | L.m | **2504** | | |
| 4.8 | Ditto, but for pipes diameter 300mm | L.m | **1867** | | |
| 4.9 | Ditto, but for pipes diameter 250mm | L.m | **1884** | | |
| 4.10 | Ditto, but for pipes diameter 200mm | L.m | **6772** | | |

### 5– concrete works

| Item | Item Description | Unit | Quantity | Unit Price € | Total amount € |
|---|---|---|---|---|---|
| 5 | **Concrete works** | | | | |
| 5.1 | Supply and cast encasement plain concrete (B-200) surround for sewer, according to drawings and specifications. | $m^3$ | 100 | | |

### 6 – Air leakage test

| Item | Item Description | Unit | Quantity | Unit Price € | Total amount € |
|---|---|---|---|---|---|
| **6** | **Air leakage test** | | | | |
| **6.1** | Air leakage test for **sewer** pipelines according to specifications, including for all temporary works | | | | |
| 6.1.1 | Nominal bore 1050mm | L.m | **2851** | | |
| 6.1.2 | Nominal bore 900mm | L.m | **2001** | | |
| 6.1.3 | Nominal bore 750mm | L.m | **2171** | | |
| 6.1.4 | Nominal bore 600mm | L.m | **3150** | | |
| 6.1.5 | Nominal bore 450mm | L.m | **1791** | | |
| 6.1.6 | Nominal bore 375mm | L.m | **2504** | | |
| 6.1.7 | Nominal bore 300mm | L.m | **1867** | | |
| 6.1.8 | Nominal bore 250mm | L.m | **1884** | | |
| 6.1.9 | Nominal bore 200mm | L.m | **6772** | | |
| **6.2** | Water leakage test for all **manholes**, according to specifications, including for all temporary works. | | | | |
| 3.2 | Ditto, but depth between 2.00-2.50m | Nr. | **169** | | |
| 3.3 | Ditto, but depth between 2.50-3.00m | Nr. | **8** | | |

### 7 – Road reinstatement

| Item | Item Description | Unit | Quantity | Unit Price € | Amount € |
|---|---|---|---|---|---|
| **7.1** | Provide and place 250 mm, base coarse For Sewer Pipes Ǿ 1050,900,750,600,450,375,300,250 & 200along with 50 mm asphalt over it, after compaction, all the work includes compaction, bitumen layer (1.0 lt./m 2) between the base coarse layer. | Lm | **8635** | | |

### Summary Table (for the project)

| | | |
|---|---|---|
| **1** | **Excavations and backfilling** | **24991** |
| **2** | **Pipes** | **24991** |
| **3** | **Concrete manholes** | **762** |
| **4** | **Pipe bedding** | **24991** |
| **5** | **Concrete works** | **100** |
| **6** | **Air leakage test** | **8635+177=8812** |
| **7** | **Road reinstatement** | **24991** |
| | **Construction of Yatta sewage network Total** | |

Discount as percentage of the total amount= ------------------------

TOTAL CONTRACT AMOUNT AFTER DISCOUNT = ------------------------

**CHAPTER**

# *5* *CONCLUSION*

---

## *CONCLUSION*

# CHAPTER FIVE
# CONCLUSIONS

The trial is made to evaluate and design wastewater collection system for Yatta town considering the annual growth of the people and their water consumption for the coming 25 years. The result brought out many important conclusions. The main conclusions drawn from the present study are summarized below:

1. Yatta town has no sewage facilities. The people are using latrines, cesspits and septic tanks. The wastewater has been seeping into the ground through the over flow of the deteriorated cesspits and latrines, causing series environmental and health problems.
2. The present population of Yatta town is around 60000 person. Prediction of the future population of Yatta town is very difficult due the political uncertainties. The rate of population growth is taken as 2.5-3.0%.
3. Restrictions on the Palestinian use of the annual ground water resources of the West Bank led to limited quantities of water supplied to Yatta town and due to this condition the average consumption of water in the camp in general is low (60 l/c.d.) and does not represent the present actual demand of water. As a result of the long period of occupation, the town lacks well studies and prepared master plan for land uses, town planning, and the design of utilities. The future water consumption is estimated to be 150 l/c.d.
4. The flow in the proposed wastewater collection system is going by gravity (gravity flow sanitary sewer system).

5. The maximum depth of flow in the sewer taken as 50% of the sewer diameter to be capable to receive any unexpected infiltration from the storm water or miss use of the sewer by the people by throughing solid waste.
6. The proposed wastewater collection system for Yatta town covers most of the areas of the town.
7. In some sewers the velocity is less than 0.6 m/s especially in the beginning of the trunk. So, flushing of the trunk is needed in the first years of usage.
8. The diameter of some pipes are not available in the market. The study recommended to use another pipes, with greater diameter.

# REFERENCES


1. Casey, T. (1992), “Water and Wastewater Engineering Hydraulics”, Oxford University Press, New York, U.S.A.
2. Hammer, Mark J. (1977), “Water and Wastewater Technology”, John Wiley and sons, INC., U.S.A.
3. Mc Ghee, Terence J. (1991), “Water Supply and Sewage “, Sixth Edition, MC Ghee- Hill International Editions, U.S.A.
4. Metcalf and Eddy, Inc. (1982), Wastewater Engineering: Treatment, Disposal, and Reuse”, Mc Grew-Hill, U.S.A.
5. Palestinian Bureau of Statistics (1999), “Locality Type Booklet: Population, Housing, and Establishment Census “, Statistical Reports Series, Ramallah, West Bank, Palestine.

# CHAPTER FIVE

# BILL OF QUANTITY
# FOR THE PROPOSED WASTEWATER COLLECTION SYSTEM

## 1 – Excavations and backfilling

| Item | Item Description | Unit | Quantity | Unit Price € | Total amount € |
|---|---|---|---|---|---|
| **1.1** | Excavations and backfilling nominal pipe diameter **1050** mm | | | | |
| 1.1.1 | Excavation of pipe trench in all kinds of soil, rock, etc, the price shall include cost of backfilling with selected suitable material approved by the engineer, and shall include leveling and compaction to a depth not exceeding 2 m and disposing surplus material outside the site according to drawings. And specifications. | L.m | **648. 5** | | |
| 1.1.2 | Ditto, but for excavations between 2.00-2.50m | L.m | **138** | | |
| 1.1.3 | Ditto, but for excavations between 2.50-4m | L.m | **2067** | | |
| **1.2** | Excavations and backfilling nominal pipe diameter **900** mm | | | | |
| 1.2.1 | Excavation of pipe trench in all kinds of soil, rock, etc, the price shall include cost of backfilling with selected suitable material approved by the engineer, and shall include leveling and compaction to a depth not exceeding 2 m and disposing surplus material outside the site according to drawings. And specifications. | L.m | **1431** | | |
| 1.2.2 | Ditto, but for excavations between 2.00-2.50m | L.m | **408.5** | | |
| 1.2.3 | Ditto, but for excavations between 2.50-4.00m | L.m | **161** | | |
| **1.3** | Excavations and backfilling nominal pipe diameter **750 mm** | | | | |
| 1.3.1 | Excavation of pipe trench in all kinds of soil, rock, etc, the price shall include cost of backfilling with selected suitable material approved by the engineer, and shall include leveling and compaction to a depth not exceeding 2 m and disposing surplus material outside the site according to drawings. And specifications. | L.m | **1465** | | |
| 1.3.2 | Ditto, but for excavations between 2.00-2.50m | L.m | **347** | | |
| 1.3.3 | Ditto, but for excavations between 2.50-4.00m | L.m | **305** | | |
| **1.4** | Excavations and backfilling nominal pipe diameter **600 mm** | | | | |
| 1.4.1 | Excavation of pipe trench in all kinds of soil, rock, etc, the price shall include cost of backfilling with selected suitable material approved by the engineer, and shall include leveling and compaction to a depth not exceeding 1.5m and disposing surplus material outside the site according to drawings. And specifications. | L.m | **1540** | | |
| 1.4.2 | Ditto, but for excavations between 2.00-2.50m | L.m | **545.5** | | |
| 1.4.3 | Ditto, but for excavations between 2.50-4 m | L.m | **1075** | | |

| Item | Item Description | Unit | Quantity | Unit Price € | Total amount € |
|---|---|---|---|---|---|
| **1.5** | Excavations and backfilling nominal pipe diameter **450** mm | | | | |
| 1.5.1 | Excavation of pipe trench in all kinds of soil, rock, etc, the price shall include cost of backfilling with selected suitable material approved by the engineer, and shall include leveling and compaction to a depth not exceeding 2 m and disposing surplus material outside the site according to drawings. And specifications. | L.m | **1791** | | |
| 1.5.2 | Ditto, but for excavations between 2.00-2.50m | L.m | | | |
| 1. 5.3 | Ditto, but for excavations between 2.50-4.00m | L.m | | | |
| **1.6** | Excavations and backfilling nominal pipe diameter **375** mm | | | | |
| 1.6.1 | Excavation of pipe trench in all kinds of soil, rock, etc, the price shall include cost of backfilling with selected suitable material approved by the engineer, and shall include leveling and compaction to a depth not exceeding 2 m and disposing surplus material outside the site according to drawings. And specifications. | L.m | **2065** | | |
| 1.6.2 | Ditto, but for excavations between 2.00-2.50m | L.m | **255** | | |
| 1.6.3 | Ditto, but for excavations between 2.50-4.00m | L.m | **32.5** | | |
| **1.7** | Excavations and backfilling nominal pipe diameter **300 mm** | | | | |
| 1.7.1 | Excavation of pipe trench in all kinds of soil, rock, etc, the price shall include cost of backfilling with selected suitable material approved by the engineer, and shall include leveling and compaction to a depth not exceeding 2 m and disposing surplus material outside the site according to drawings. And specifications. | L.m | **1810** | | |
| 1.7.2 | Ditto, but for excavations between 2.00-2.50m | L.m | **28** | | |
| 1.7.3 | Ditto, but for excavations between 2.50-4.00m | L.m | | | |
| **1.8** | Excavations and backfilling nominal pipe diameter **250 mm** | | | | |
| 1.8.1 | Excavation of pipe trench in all kinds of soil, rock, etc, the price shall include cost of backfilling with selected suitable material approved by the engineer, and shall include leveling and compaction to a depth not exceeding 2 m and disposing surplus material outside the site according to drawings. And specifications. | L.m | **1857** | | |
| 1.8.2 | Ditto, but for excavations between 2.00-2.50m | L.m | | | |
| 1.8.3 | Ditto, but for excavations between 2.50-4.00m | L.m | | | |
| **1.9** | Excavations and backfilling nominal pipe diameter **200 mm** | | | | |
| | Excavation of pipe trench in all kinds of soil, rock, etc, the price shall include cost of backfilling with selected suitable material approved by the engineer, and shall include leveling and compaction to a depth not exceeding 2 m and disposing surplus material outside the site according to drawings. And specifications. | Lm | **6730** | | |
| 1.9.1 | Ditto, but for excavations between 2.00-2.50m | Lm | **198. 5** | | |
| 1.9.2 | Ditto, but for excavations between 2.50-4.00m | Lm | **251. 5** | | |

## 2– Pipes

| Item | Item Description | Unit | Quantity | Unit Price € | Total amount € |
|---|---|---|---|---|---|
| | **Pipes** | | | | |
| **2.1** | Supply, store and installation of pipes diameter **1050(uPVC)**with the Techen stamp or equivelant, along with the fittings, according to drawings, and specifications. | L.m | **2851** | | |
| 2.2 | Ditto, but for pipes diameter **900mm(u PVC)** | L.m | **2001** | | |
| 2.3 | Ditto, but for pipes diameter **750mm(u PVC)** | L.m | **2171** | | |
| 2.4 | Ditto, but for pipes diameter **600mm(u PVC)** | L.m | **3150** | | |
| 2.5 | Ditto, but for pipes diameter **450mm(u PVC)** | L.m | **1791** | | |
| 2.6 | Ditto, but for pipes diameter **375mm(u PVC)** | L.m | **2504** | | |
| 2.7 | Ditto, but for pipes diameter **300mm(u PVC)** | L.m | **1867** | | |
| 2.8 | Ditto, but for pipes diameter **250mm(u PVC)** | L.m | **1884** | | |
| 2.9 | Ditto, but for pipes diameter **200mm(u PVC)** | L.m | **6772** | | |

## 3 – Concrete manholes

| Item | Item Description | Unit | Quantity | Unit Price € | Total amount € |
|---|---|---|---|---|---|
| **3** | **Concrete manholes**<br>**Precast manholes** | | | | |
| 3.1 | Supplying and installation of manhole, coated with coal tar epoxy, including excavations in all kinds of soil, rock, etc, shall include cost of backfilling with selected suitable material approved by the engineer, and steps and benching, heavy duty cover 25 tons for streets, and 8 tons for cross country fields and backfilling not exceeding 2m .**Diameter1200mm** according to drawings and specifications. | Nr. | **107** | | |
| 3.2 | Ditto, but depth between 2.00-2.50m | Nr. | **393** | | |
| 3.3 | Ditto, but depth between 2.50-4.00m | Nr. | **262** | | |

## 4 – Pipe bedding

| Item | Item Description | Unit | Quantity | Unit Price € | Total amount € |
|---|---|---|---|---|---|
| **4** | **Pipe bedding** | | | | |
| 4.1 | Supplying, installation and compaction of (**Absolet**) fine granular material, under, above and around pipe Diameter (according to depth at items 1.1) **1050mm** according to the drawings and specifications. | L.m | **2851** | | |
| 4.2 | Ditto, but for pipes diameter 900mm | L.m | **2001** | | |
| 4.3 | Ditto, but for pipes diameter 750mm | L.m | **2171** | | |
| 4.5 | Ditto, but for pipes diameter 600mm | L.m | **3150** | | |
| 4.6 | Ditto, but for pipes diameter 450mm | L.m | **1791** | | |
| 4.7 | Ditto, but for pipes diameter 375mm | L.m | **2504** | | |
| 4.8 | Ditto, but for pipes diameter 300mm | L.m | **1867** | | |
| 4.9 | Ditto, but for pipes diameter 250mm | L.m | **1884** | | |
| 4.10 | Ditto, but for pipes diameter 200mm | L.m | **6772** | | |

### 5– concrete works

| Item | Item Description | Unit | Quantity | Unit Price € | Total amount € |
|---|---|---|---|---|---|
| 5 | **Concrete works** | | | | |
| 5.1 | Supply and cast encasement plain concrete (B-200) surround for sewer, according to drawings and specifications. | $m^3$ | 100 | | |

### 6 – Air leakage test

| Item | Item Description | Unit | Quantity | Unit Price € | Total amount € |
|---|---|---|---|---|---|
| **6** | **Air leakage test** | | | | |
| **6.1** | Air leakage test for **sewer** pipelines according to specifications, including for all temporary works | | | | |
| 6.1.1 | Nominal bore 1050mm | L.m | **2851** | | |
| 6.1.2 | Nominal bore 900mm | L.m | **2001** | | |
| 6.1.3 | Nominal bore 750mm | L.m | **2171** | | |
| 6.1.4 | Nominal bore 600mm | L.m | **3150** | | |
| 6.1.5 | Nominal bore 450mm | L.m | **1791** | | |
| 6.1.6 | Nominal bore 375mm | L.m | **2504** | | |
| 6.1.7 | Nominal bore 300mm | L.m | **1867** | | |
| 6.1.8 | Nominal bore 250mm | L.m | **1884** | | |
| 6.1.9 | Nominal bore 200mm | L.m | **6772** | | |
| **6.2** | Water leakage test for all **manholes**, according to specifications, including for all temporary works. | | | | |
| 3.2 | Ditto, but depth between 2.00-2.50m | Nr. | **169** | | |
| 3.3 | Ditto, but depth between 2.50-3.00m | Nr. | **8** | | |

### 7 – Road reinstatement

| Item | Item Description | Unit | Quantity | Unit Price € | Amount € |
|---|---|---|---|---|---|
| **7.1** | Provide and place 250 mm, base coarse For Sewer Pipes Ó 1050,900,750,600,450,375,300,250 & 200along with 50 mm asphalt over it, after compaction, all the work includes compaction, bitumen layer (1.0 lt./m 2) between the base coarse layer. | Lm | **8635** | | |

### Summary Table (for the project)

| | | |
|---|---|---|
| **1** | **Excavations and backfilling** | **24991** |
| **2** | **Pipes** | **24991** |
| **3** | **Concrete manholes** | **762** |
| **4** | **Pipe bedding** | **24991** |
| **5** | **Concrete works** | **100** |
| **6** | **Air leakage test** | **8635+177=8812** |
| **7** | **Road reinstatement** | **24991** |
| | **Construction of Yatta sewage network Total** | |

Discount as percentage of the total amount= ------------------------

TOTAL CONTRACT AMOUNT AFTER DISCOUNT = ------------------------

USER'S GUIDE

# SewerCAD v5

**for Windows**

SewerCAD v5 User's Guide (Second Printing)



**Trademarks**



37 Brookside Rd.

Waterbury, CT 06708-1499

Voice: +1 - (203) 755-1666

FAX: +1 - (203) 597-1488

e-mail: info@haestad.com

Internet: http://www.haestad.com

# Table of Contents

**Preface** **xiii**

**Welcome to SewerCAD** **xiii**

**Chapter 1 Orientation** **1**

**1.1 What is SewerCAD?** **1**

**1.2 When to Use SewerCAD?** **1**

**1.3 What's New in SewerCAD?** **2**

**1.4 Installation, Upgrades and Updates** **3**

1.4.1 Minimum System Requirements 3
1.4.2 Installing Haestad Methods' Products 4
1.4.3 Uninstalling Haestad Methods' Products 4
1.4.4 Troubleshooting Setup or Uninstall 4
1.4.5 Software Registration 5
1.4.6 Upgrades 5
1.4.7 Globe Button 5
1.4.8 Network Licensing 6

**1.5 Learning SewerCAD** **9**

1.5.1 SewerCAD Documentation 9
1.5.2 How to Use Help 9
1.5.3 How Do I? 9
1.5.4 Glossary 10
1.5.5 Tutorials 10
1.5.6 Sample Projects 10
1.5.7 Haestad Methods Workshops and Certification 10

**1.6 Contacting Haestad Methods** **11**

1.6.1 Sales 11
1.6.2 Technical Support 11
1.6.3 Your Suggestions Count 12
1.6.4 How to Contact Us 12

**Chapter 2 SewerCAD Main Window** **13**

**2.1 Overview** **13**

**2.2 Main Window Components** **13**

2.2.1 Stand-Alone Mode, AutoCAD Mode 13
2.2.2 SewerCAD Main Windows 13
2.2.3 Drawing Pane 15
2.2.4 Status Pane 15
2.2.5 Menus, Toolbars, and Shortcut Keys 16
2.2.6 Command Line 16

**2.3 SewerCAD Menus** **17**

2.3.1 Pull-down Menus 17
2.3.2 File Menu 17
2.3.3 Edit Menu 19
2.3.4 Analysis Menu 21
2.3.5 View Menu 21
2.3.6 Tools Menu 22

2.3.7 Report Menu 24
2.3.8 Draw Menu (in AutoCAD Mode Only) 25
2.3.9 Help Menu 25

**2.4 SewerCAD Toolbars 26**
2.4.1 Toolbar Button Summaries 26
2.4.2 Tool Pane Summary 26
2.4.3 SewerCAD Tool Palette 28
2.4.4 Analysis Toolbar 28
2.4.5 SewerCAD VCR Controls 28
2.4.6 Other Toolbar Buttons 29

**2.5 The Status Bar 30**
2.5.1 General Status Information 30
2.5.2 DXF Background Status 30
2.5.3 Cursor Location 30
2.5.4 Calculation Results Status 31
2.5.5 File Status 31

## Chapter 3 Quick Start Lessons 33

**3.1 Overview 33**

**3.2 Lesson 1 - Creating a Schematic Network 33**

**3.3 Lesson 2 - Automatic Design 43**

**3.4 Lesson 3 - Scenario Management 45**

**3.5 Lesson 4 - Presentation of Results 50**

**3.6 Lesson 5 - Running an Extended Period Simulation 56**

## Chapter 4 Starting a SewerCAD Project 63

**4.1 Overview 63**

**4.2 File Management 63**
4.2.1 Multiple Sessions 63

**4.3 Project Management 63**
4.3.1 Project Setup Wizard 63
4.3.2 Project Summary 64

**4.4 Options 64**
4.4.1 Global Options 64
4.4.2 Project Options 66
4.4.3 Drawing Options 67

**4.5 FlexUnits 69**
4.5.1 FlexUnits Overview 69
4.5.2 Field Options 69
4.5.3 Units 70
4.5.4 Display Precision 70
4.5.5 Scientific Notation 70
4.5.6 Minimum and Maximum Allowed Value 70
4.5.7 FlexUnits Manager 71

**4.6 Quick Attribute Selector 71**

## Chapter 5 Layout and Editing Tools 73

**5.1 Graphical Editor Overview 73**

**5.2 Graphical Editor 73**
5.2.1 Using the Graphical Editor 73
5.2.2 Working with Network Elements Within the Graphical Editor 73
5.2.3 Creating New Elements 74
5.2.4 Changing the Pipe Layout Tool to Insert a Different Type of Node 74
5.2.5 Morphing Elements 74
5.2.6 Splitting Pipes 75
5.2.7 Selecting Elements 75
5.2.8 Editing Elements 76
5.2.9 Moving Elements 76
5.2.10 Deleting Elements 77
5.2.11 Other Tools 77

**5.3 Selection Sets 77**
5.3.1 Selection Set Manager 78
5.3.2 New Selection Set 78
5.3.3 Selection Set Dialog 78
5.3.4 Duplicate Selection Set 78
5.3.5 Delete Selection Set 78
5.3.6 Rename Selection Set 78
5.3.7 Selection Set Notes 78

**5.4 Find Element 78**

**5.5 Zooming 79**
5.5.1 Zoom Center 79
5.5.2 Aerial View 80

**5.6 Drawing Review 80**
5.6.1 Selection Tolerance 81

**5.7 Relabel Elements 82**
5.7.1 Relabel Operations 82
5.7.2 Elements Selected 83

**5.8 Element Labeling 83**

**5.9 Quick View 84**

**Chapter 6 Hydraulic Element Editors 85**

**6.1 Overview 85**

**6.2 Element Editors 86**
6.2.1 Using Element Editors 86
6.2.2 Manholes 86
6.2.3 Junction Chambers 87
6.2.4 Wet Wells 87
6.2.5 Pumps 87
6.2.6 Pressure Junctions 88
6.2.7 Outlets 88
6.2.8 Gravity Pipes 88
6.2.9 Pressure Pipes 89

**6.3 Element Editors' Tabs 89**
6.3.1 General Tab 89
6.3.2 Headlosses Tab 99
6.3.3 Diversion Tab 100
6.3.4 Controls Tab 101
6.3.5 Profile Tab 103
6.3.6 Design Tab 104
6.3.7 Section Tab 105

6.3.8 Loading Tab 107
6.3.9 Infiltration Tab 108
6.3.10 Cost Tab 109
6.3.11 User Data Tab 110
6.3.12 Message Tab 111

**6.4 Loading Dialogs 111**
6.4.1 Add New Load Dialog 111
6.4.2 Base Load Dialog 112
6.4.3 Hydrograph Dialog 112
6.4.4 Pattern Load Dialog 112

**6.5 Prototypes 113**

**6.6 User Data Extensions 113**
6.6.1 User Data Extensions Dialog 113

**Chapter 7 FlexTables 117**

**7.1 Tabular Reporting Overview 117**

**7.2 Table Manager 117**
7.2.1 Creating New Tables 118
7.2.2 Two Row Tables 118
7.2.3 Editing Tables 118
7.2.4 Duplicating Tables 118
7.2.5 Deleting Tables 118
7.2.6 Renaming Tables 118
7.2.7 Resetting Tables 119

**7.3 Table Setup Dialog 119**
7.3.1 Table Type 119
7.3.2 Available Table Columns 120
7.3.3 Selected Table Columns 120
7.3.4 Table Manipulation Buttons 120
7.3.5 Allow Duplicate Columns 120

**7.4 Table Window 120**
7.4.1 Editing Tables 121
7.4.2 Sorting/Filtering Tables 122
7.4.3 Table Customization 124
7.4.4 Table Output 126

**Chapter 8 Scenarios and Alternatives 127**

**8.1 Overview 127**

**8.2 Alternatives 127**
8.2.1 Alternatives Manager 128
8.2.2 Alternatives Editor 129
8.2.3 Physical Properties Alternative Editor 130
8.2.4 Sanitary (Dry Weather) Loading Alternative Editor 133
8.2.5 Infiltration and Inflow Loading Alternative Editor 134
8.2.6 Known Flow Loading Alternative Editor 135
8.2.7 Structure Headlosses Alternative Editor 135
8.2.8 Boundary Conditions Alternative Editor 136
8.2.9 Design Constraints Alternative Editor 136
8.2.10 Initial Settings Alternative Editor 138
8.2.11 Operational Alternative Editor 139
8.2.12 Cost Alternative Editor 139

8.2.13 User Data Alternative Editor 139

**8.3 Scenarios 139**
8.3.1 Scenario Selection 140
8.3.2 Editing Scenarios 140
8.3.3 Scenario Manager 140
8.3.4 Scenario Wizard 142
8.3.5 Scenario Editor 144

## Chapter 9 Modeling and Design Capabilities 147

**9.1 Calculate 147**
9.1.1 Calculation Type Section 147
9.1.2 Steady State Section 147
9.1.3 Extended Period Section 148

**9.2 Calculations Options 148**
9.2.1 Gravity Hydraulics Tab 148
9.2.2 HEC-22 Tab 149
9.2.3 AASHTO Tab 150
9.2.4 Generic Structure Loss Tab 150
9.2.5 Convex Routing Tab 151
9.2.6 Steady State Loading Options Tab 151
9.2.7 Pressure Hydraulics Tab 151

**9.3 Pattern Manager 152**
9.3.1 Patterns 152
9.3.2 Pattern Manager 153
9.3.3 Pattern Editor 154
9.3.4 Pattern Graph and Report 155

**9.4 Pattern Setup Manager 155**
9.4.1 Pattern Setup Editor 155

**9.5 Extreme Flow Setup Manager 156**
9.5.1 Extreme Flow Setup Editor 156

**9.6 Default Design Constraints 157**
9.6.1 Gravity Pipe Tab 157
9.6.2 Default Constraints Section 157
9.6.3 Extended Design Section 157
9.6.4 Gravity Structure Tab 158

## Chapter 10 Cost Estimating 159

**10.1 Overview 159**

**10.2 Cost Manager 159**
10.2.1 Cost Manager - Button Section 160
10.2.2 Cost Manager - Center Pane 160
10.2.3 Cost Manager - Left Pane 161
10.2.4 System Cost Adjustments Table 161

**10.3 Unit Cost Functions 161**
10.3.1 Unit Cost Functions Manager 161
10.3.2 Tabular Unit Cost Function 162
10.3.3 Formula Unit Cost Function 162
10.3.4 Unit Cost Function Notes 163

**10.4 Cost Alternatives Manager 163**

**10.5 Cost Reports 164**

10.5.1 Element Detailed Cost Report 164
10.5.2 Project Detailed Cost Report 165
10.5.3 Project Element Summary Cost Report 165
10.5.4 Project Summary Cost Report 165
10.5.5 Pipe Costs Report 165
10.5.6 Cost Warnings Report 165

**Chapter 11 Presenting your Results 167**

**11.1 Overview 167**

**11.2 Element Annotation 167**
11.2.1 Attribute Annotation Dialog 167
11.2.2 Annotation Properties 168
11.2.3 The Annotation Wizard 168

**11.3 Color Coding 169**
11.3.1 Color Coding Dialog 170

**11.4 Reporting 170**
11.4.1 Predefined Reports 170
11.4.2 Element Details Report 171
11.4.3 Element Results Report 171
11.4.4 Tabular Reports 172
11.4.5 Scenario Summary Report 172
11.4.6 Project Inventory Report 172
11.4.7 Plan View Report 172
11.4.8 Calculation / Problem Summary Report 173

**11.5 Graphing 173**
11.5.1 Graph Setup 173
11.5.2 Graph Window 174
11.5.3 Plot Window 174
11.5.4 Graph Options 174

**11.6 Pie Charts 176**

**11.7 Profile 176**
11.7.1 Profile Manager 176
11.7.2 Profile Templates 177
11.7.3 Profile Wizard 179
11.7.4 Profile Window 180

**11.8 Diversion Network 183**
11.8.1 Diversion Network Window 183
11.8.2 Diversion Network Options 184
11.8.3 Diversion Network Background Color 184

**11.9 Scenario Comparison 184**
11.9.1 Annotation Comparison Wizard 184
11.9.2 Scenario Comparison Window 184

**11.10 Graphic Annotation 185**
11.10.1 Legend 186

**11.11 Preview Windows 186**

**11.12 Status Log 187**

**Chapter 12 Engineering Libraries 189**

**12.1 Engineering Libraries Overview 189**

**12.2 Engineering Library Manager 189**

**12.3 Engineering Library Editor 190**

**12.4 Usage 191**

**12.5 Extreme Flow Factor Method Library 191**
12.5.1 Extreme Flow Factor Equation Properties 191
12.5.2 Extreme Flow Factor Table Properties 192

**12.6 Section Size 192**
12.6.1 Section Size Library 192
12.6.2 Arch Section Size Properties 193
12.6.3 Box Section Size Properties 193
12.6.4 Circular Section Size Properties 194
12.6.5 Ellipse Section Size Properties 194
12.6.6 Available in Materials 194

**12.7 Material Properties 195**
12.7.1 Material Library 195

**12.8 Minor Loss 195**
12.8.1 Minor Loss Properties 195

**12.9 Unit Sanitary (Dry Weather) Load 196**
12.9.1 Population-Based Unit Sanitary (Dry Weather) Load Properties 196
12.9.2 Non-Population-Based Unit Sanitary (Dry Weather) Loads 196
12.9.3 Area-Based Unit Sanitary (Dry Weather) Loads 197
12.9.4 Discharge-Based Unit Sanitary (Dry Weather) Loads 197
12.9.5 Count-Based Unit Sanitary (Dry Weather) Loads 197

**Chapter 13 GIS and Database Connections 199**

**13.1 Overview 199**

**13.2 Database Connections 201**
13.2.1 Database Connection Manager 201
13.2.2 Standard Database Import/Export 201
13.2.3 Database Connection Editor 203
13.2.4 ODBC 206
13.2.5 Sharing Database Connections between Projects 207
13.2.6 Database Connection Example 208

**13.3 Shapefile Connections 208**
13.3.1 Shapefile Connection Manager 208
13.3.2 Shapefile Connection Editor 209
13.3.3 Shapefile Link Wizard 210
13.3.4 Import Shapefile Wizard 211
13.3.5 Export Shapefile Wizard 213
13.3.6 Sharing Shapefile Connections between Projects 214
13.3.7 Shapefile Format 215
13.3.8 Shapefile Connection Example 215

**Chapter 14 Exchanging Data with CAD Software and Autodesk Civil Design 217**

**14.1 AutoCAD Polyline to Pipe Conversion 217**
14.1.1 Polyline to Pipe Wizard 218
14.1.2 Polyline to Pipe Wizard - Step 1 (Stand-Alone mode only) 218
14.1.3 Polyline to Pipe Wizard - Step 2 218
14.1.4 Polyline to Pipe Wizard - Step 3 219

14.1.5 Polyline to Pipe Wizard - Step 4 (for .DXF files that contain blocks) 219
14.1.6 Polyline to Pipe Wizard - Step 5 220
14.1.7 Polyline to Pipe Wizard - Step 6 220
14.1.8 Drawing Preview 220
14.1.9 Polyline Conversion Problem Dialog 220
14.1.10 Converting your Drawing in Multiple Passes 220

**14.2 Land Development Desktop - Civil Design Connection 220**
14.2.1 Land Development Desktop Import Wizard 220
14.2.2 File Import Settings 221
14.2.3 Runs to Import 221
14.2.4 Import Structure Mappings 221
14.2.5 Land Development Desktop Export Wizard 221
14.2.6 File Export Settings 222
14.2.7 Runs to Export 222
14.2.8 Add/Edit Pipe Run 222
14.2.9 Delete Runs 223
14.2.10 Initialize Run List 223
14.2.11 Automatic Element Labeling 223
14.2.12 Export Structure Mapping 223

**14.3 Import/Export of DXF Files 223**
14.3.1 Import a DXF from AutoCAD or MicroStation 223
14.3.2 Exporting a DXF file 223
14.3.3 Redefining SewerCAD Blocks in AutoCAD 224
14.3.4 Advanced DXF Import Techniques 224

**Chapter 15 Additional Features of the AutoCAD Version 225**

**15.1 Overview 225**

**15.2 SewerCAD Custom AutoCAD Entities (AutoCAD Mode) 225**

**15.3 AutoCAD Environment 226**
15.3.1 AutoCAD Mode Graphical Layout 226
15.3.2 Toolbars 226
15.3.3 Drawing Setup 227
15.3.4 Symbol Visibility 227
15.3.5 Rebuild Figure Labels 227

**15.4 AutoCAD Project Files 227**
15.4.1 Drawing Synchronization 228
15.4.2 Saving the Drawing as Drawing*.dwg 228

**15.5 Element Properties 228**
15.5.1 Select Layer 229
15.5.2 Select Text Style 229

**15.6 Working with Elements 229**
15.6.1 Edit Element 229
15.6.2 Edit Elements 229
15.6.3 Deleting Elements 229
15.6.4 Modifying Elements 229

**15.7 Working with Elements Using AutoCAD Commands 230**
15.7.1 AutoCAD Commands 230
15.7.2 Explode Elements 230
15.7.3 Moving Elements 230
15.7.4 Moving Element Labels 231
15.7.5 Snap Menu 231

**15.8 Undo / Redo 231**

15.8.1 Undo and Redo Operations in AutoCAD 231

**15.9 Converting Native AutoCAD Entities to SewerCAD Elements 231**
15.9.1 Converting Native AutoCAD Entities 231
15.9.2 Layout Pipe Using Entity 232
15.9.3 Change Entities to Pipes 232

**15.10 Special Considerations 232**
15.10.1 Import SewerCAD 232
15.10.2 Working with Proxies 233

**Appendix A Frequently Asked Questions 235**

**A .1 Overview 235**

**A .2 How Do I Control Element and Label Sizing? 235**

**A .3 How Do I Reuse Deleted Element Labels? 235**

**A .4 How Do I Color Code Elements? 235**

**A .5 How Do I Remove Color Coding from Labels Imported from Pre-v3.5 AutoCAD Files? 236**

**A .6 How Do I Do a Profile Plot? 236**

**A .7 How Do I Change Units in a Column? 236**

**A .8 How Do I Access the Haestad Methods Knowledge Base? 237**

**A .9 How do I Model an Inverted Siphon (Depressed Sewer)? 237**

**Appendix B SewerCAD Theory 239**

**B .1 Overview 239**

**B .2 Loading 239**
B.2.1 Common Load Types 240
B.2.2 Sanitary (Dry Weather) Loading 241
B.2.3 Wet Weather Loading 242
B.2.4 Known Loading 243

**B .3 Gravity Pipe Hydraulics 244**
B.3.1 Basic Concepts 244
B.3.2 Hydraulics and Energy Grades 244
B.3.3 Friction Loss Methods 245
B.3.4 Flow Regime 249
B.3.5 Gradually Varied Flow Analysis 250
B.3.6 Energy Balance 252
B.3.7 Mixed Flow Profiles 253
B.3.8 Backwater Analysis 254
B.3.9 Frontwater Analysis 254
B.3.10 Pipe Average Velocity 255
B.3.11 Capacity Analysis (Approximate Profiles) 256

**B .4 Junction Headlosses and Minor Losses 258**
B.4.1 Junction Headlosses 258
B.4.2 Minor Losses 266

**B .5 Pumping Stations and Pressure Sewers 267**
B.5.1 Pump Theory 267
B.5.2 Pump Type 269
B.5.3 Conservation of Mass and Energy 270
B.5.4 The Gradient Algorithm 270
B.5.5 Derivation of the Gradient Algorithm 271

| | | | |
|---|---|---|---|
| | B.5.6 | The Linear System Equation Solver | 273 |
| **B .6** | **Extended Period Simulations** | | **274** |
| | B.6.1 | Extended Period Simulations Overview | 274 |
| | B.6.2 | Routing Overview | 274 |
| | B.6.3 | Convex Routing | 274 |
| | B.6.4 | Weighted Translation Routing | 275 |
| | B.6.5 | Hydrologic and Hydraulic Time Steps | 276 |
| **B .7** | **Transitioning between Gravity and Pressure Networks** | | **276** |
| | B.7.1 | Overview | 276 |
| | B.7.2 | Identifying Gravity Pipes and Force Mains | 276 |
| | B.7.3 | Direction of Flow in Gravity and Pressure Systems | 276 |
| | B.7.4 | Transitioning From Gravity Pipes to Force Mains | 277 |
| | B.7.5 | Transitioning From Force Mains to Gravity Elements | 277 |
| **B .8** | **Constraint Based Automatic Design** | | **280** |
| | B.8.1 | Gravity Pipes and Structures Design | 280 |
| | B.8.2 | Part Full Design | 281 |
| | B.8.3 | Allow Multiple Sections | 281 |
| | B.8.4 | Limit Section Size | 282 |
| | B.8.5 | Pipe Matching | 282 |
| | B.8.6 | Offset Matching | 282 |
| | B.8.7 | Drop Structures | 283 |
| | B.8.8 | Structure Sump Elevations | 283 |
| | B.8.9 | Design Priorities | 283 |
| | B.8.10 | Automatic Design with Hydrograph and Pattern Loads | 285 |
| | B.8.11 | Constraint Based Warning Messages | 286 |
| **B .9** | **Special Considerations** | | **286** |
| | B.9.1 | Energy Discontinuity | 286 |
| | B.9.2 | Structure Energy Grade | 286 |
| | B.9.3 | Design Considerations | 287 |
| | B.9.4 | Reporting Flow Attributes | 287 |
| **B .10** | **Engineer's Reference** | | **287** |
| | B.10.1 | Default Kinematic Viscosity | 288 |
| | B.10.2 | Headloss Coefficients for Junctions | 289 |
| | B.10.3 | Roughness Values - Manning's Equation | 290 |
| | B.10.4 | Roughness Values - Kutter’s Equation | 290 |
| | B.10.5 | Roughness Values - Darcy-Weisbach Equation (Colebrook-White) | 291 |
| | B.10.6 | Roughness Values - Hazen-Williams Formula | 292 |
| | B.10.7 | Typical Roughness Values for Pressure Pipes | 293 |

**Appendix C Importing Loading Data 295**

| | | | |
|---|---|---|---|
| **C .1** | **Importing Loading Data Overview** | | **295** |
| **C .2** | **Import Loading Data Dialog** | | **295** |
| **C .3** | **Loading Data Text File Format** | | **296** |
| | C.3.1 | ASCII Loading Data Format | 296 |
| | C.3.2 | Constant Increment Patterns Section | 296 |
| | C.3.3 | Variable Increment Patterns Section | 296 |
| | C.3.4 | Sanitary Pattern Loads Section | 297 |
| | C.3.5 | Wet Pattern Loads Section | 297 |
| | C.3.6 | Constant Increment Hydrographs Section | 298 |
| | C.3.7 | Variable Increment Hydrographs Section | 298 |
| | C.3.8 | Sanitary Hydrograph Loads Section | 299 |
| | C.3.9 | Wet Hydrograph Loads Section | 299 |

C.3.10 Sanitary Unit Loads Section 300
C.3.11 Options Section 300
C.3.12 ASCII Loading Data Example 301

## Appendix D Scenario Management Reference Guide 303

**D .1 Overview 303**
**D .2 About this Guide 303**
**D .3 Before Haestad Methods: Distributed Scenarios 304**
**D .4 With Haestad Methods: Self-Contained Scenarios 305**
**D .5 The Scenario Cycle 305**
**D .6 Scenario Anatomy: Attributes and Alternatives 306**
**D .7 A Familiar Parallel 306**
**D .8 Scenario Behavior: Inheritance 307**
**D .9 Overriding Inheritance 307**
**D .10 Dynamic Inheritance 308**
**D .11 When are values local, and when are they inherited? 308**
**D .12 Minimizing Effort through Attribute Inheritance 308**
**D .13 Minimizing Effort through Scenario Inheritance 309**
**D .14 A Water Distribution Example 310**
**D .15 Building the Model (Average Day Conditions) 310**
**D .16 Analyzing Different Demands (Maximum Day Conditions) 311**
**D .17 Another Set of Demands (Peak Hour Conditions) 311**
**D .18 Correcting an Error 312**
**D .19 Analyzing Improvement Suggestions 312**
**D .20 Finalizing the Project 313**
**D .21 Summary 313**
**D .22 Conclusion 314**

## Appendix E Haestad Methods Software 315

**E .1 Overview 315**
**E .2 WaterCAD 315**
**E .3 SewerCAD 315**
**E .4 StormCAD 316**
**E .5 PondPack 316**
**E .6 CulvertMaster 316**
**E .7 FlowMaster 316**

**Glossary** 317

**References** 333

**Index** 335

# Preface

## Welcome to SewerCAD

Thank you for purchasing SewerCAD. At Haestad Methods, we pride ourselves in providing the very best engineering software available. Our goal is to make software that is easy to install and use, yet so powerful and intuitive that it anticipates your needs without getting in your way.

SewerCAD is a feature-rich program with extensive on-line documentation that is able to provide a level of instruction appropriate to your needs. Do not be fooled by the existence of this user's guide. You do not need to read anything to get started!

When you first use the program, SewerCAD's intuitive interface and interactive dialogs will guide you. If you need more information, go to our on-line help by simply pressing the **F1** key anywhere in the program. Help text regarding the area of the program in which you are working will be displayed.

We are betting that you will be able to use our product right out of the package. If you know how to run Setup within Windows, then go ahead and get right to work - install SewerCAD, and enjoy!

Notes

# Chapter 1

# Orientation

## 1.1 What is SewerCAD?

SewerCAD is an extremely powerful program for the design and analysis of gravity flow and pressure flow through pipe networks and pumping stations. The program can be run in AutoCAD mode, giving you all the power of AutoCAD's capabilities, or in Stand-Alone mode utilizing our own graphical interface. SewerCAD allows you to construct a graphical representation of a pipe network containing information such as pipe data, pump data, loading, and infiltration. You have a choice of conveyance elements including circular pipes, arches, boxes and more.

The gravity network is calculated using the built-in numerical model, which utilizes both the direct step and standard step gradually varied flow methods. Flow calculations are valid for both surcharged and varied flow situations, including hydraulic jumps, backwater, and drawdown curves. You also have the flexibility to mix gravity and pressure components freely, building your systems in parallel or in series as they exist in the field. Pressure elements can be controlled based on system hydraulics, turning pumps on and off due to changes in flows and pressures.

SewerCAD's flexible reporting feature allows you to customize and print the model results in both a report format and as a graphical plot.

## 1.2 When to Use SewerCAD?

SewerCAD is so flexible you can use it for all phases of your project, from the feasibility report to the final design drawings and analysis of existing networks. During the feasibility phase, you can use SewerCAD to create several different system layouts with an AutoCAD or MicroStation drawing as the background, or within AutoCAD itself. For the final design, you can complete detailed drawings with notes that can be used to develop the construction plans. In summary, you can use SewerCAD to:

- Design multiple sanitary sewer systems.
- Analyze various design scenarios for sanitary sewer systems.
- Import and export AutoCAD and MicroStation .DXF files.
- Generate professional-looking reports for clients.
- Generate plan and profile plots of a network.

## 1.3 What's New in SewerCAD?

### *What's New in SewerCAD 5.0*

- **Extended Period Simulations** - Run time-based simulations routing hydrographs, fixed and pattern loads through complex sanitary networks containing both gravity and pressure elements.
- **Element Graphing and Tables -** Generate graphs and tables comparing information for different hydraulic elements as well as different scenarios over time.
- **Diurnal Loading Patterns** - Apply diurnal patterns to fixed loads or to unit sanitary loads.
- **Loading Data Import -** Import multiple unit sanitary loads, wet weather loads, hydrographs and diurnal patterns from a formatted ASCII text file.
- **Panning -** Scroll easily through a SewerCAD drawing using your mouse with new panning support.
- **Profile Customization -** Create profiles with custom annotations, layer properties, and direction in order to mimic all design drawing and master plan requirements.
- **Persistent Profiles -** Store unlimited different customized profile views using the **Profiles Manager** within the same network model.
- **Profile Templates -** Create templates that can be applied to new profiles. The templates will then establish the annotations and text properties for the new profile, instead of having to reestablish all the information.
- **Editable Quick View –** Edit data efficiently through the **Quick View** window without having to open an element's dialog.
- **Wet Well Alarm Levels -** Trigger a warning message or a change in color coding when the water surface elevation in a wet well during an analysis goes above a user-specified alarm level.
- **Improved Results Reporting -** View calculation results in an intuitive directory structure. Warning and error messages are organized in a single printable table.
- **Animated Displays -** Animate the plan and profile views separately or at the same time to serve as an ideal tool for presentations and output analysis.
- **Quick Attribute Selector -** Select attributes for annotations, color coding, tables, and database / GIS connections from logically organized categories.
- **Mouse Wheel Support -** Pan and zoom using the mouse wheel.
- **Microsoft Office XP Support** - Take advantage of the capabilities of Microsoft Excel XP and Access XP for cut-copy-and-paste, as well as database integration.
- **Windows XP Support -** Load the current version of SewerCAD on Windows 95, Windows 98, NT 4.0, Me, Windows 2000, and Windows XP.

### *What's New in SewerCAD 4.1.1*

- **Diversions and Overflows -** Model CSO and SSO's, flow splits, basement flooding, or any other situation where a portion of flow needs to be removed from a gravity system.
- **Sticky Pipe Inverts -** Attach gravity pipe inverts to the sump of the adjacent element or specify the pipe inverts independently. If you select the "sticky pipe invert" mode, pipe inverts will move as you adjust the element sump elevation.
- **Minimum Structure Headloss -** Specify a global minimum structure headloss as required by many regulatory agencies.

- **Generic Headloss Method -** Provides the flexibility needed to accommodate any headloss methodology that relies on both upstream and downstream velocity heads.

*What's New in SewerCAD 4.1*

- **Network Licensing -** Purchase a multi-seat license. With the purchase of the AutoCAD version of SewerCAD, your engineers and technicians can individually use SewerCAD in either Stand-Alone mode or AutoCAD mode and share project files.
- **Performance Optimization** - Experience a 150% performance increase in Calculation Engine, 200% increase in Model Validation, 100% increase in Graphical Editor, and 200% performance increase in opening and saving files.
- **Persistent Sorts and Filters** - Maintain user-defined sorts and filters each time a table is opened.
- **Appearance of Drawing Environment** - Enjoy total control over background and foreground colors in the graphical editor.
- **Aerial View** - Access this separate window to facilitate zooming, panning, and locating a small viewing area in the main window.
- **Cost Estimating -** Perform detailed cost estimates using an integrated cost analysis modeling subsystem.
- **Extensive Documentation** - Access our extensive Internet-based support database, called the KnowledgeBase™, through our ClientCare™ Program.
- **Technical Support** - Subscribe to one of our ClientCare™ packages and access technical support seven days a week.

# 1.4 Installation, Upgrades and Updates

## 1.4.1 Minimum System Requirements

Below are the minimum and recommended system requirements for running SewerCAD with acceptable performance. Some of the requirements for AutoCAD Mode, such as RAM, are fairly high due to AutoCAD and operating system demands, not SewerCAD itself.

Stand-Alone Mode:

| | |
|---|---|
| Processor: | Pentium-166 |
| RAM: | 32 Megabytes |
| Hard Disk: | 25 Megabytes of free storage space, plus room for data files |
| Display: | 800 x 600 resolution, 256 colors |

AutoCAD Mode:

| | |
|---|---|
| Processor: | Pentium-166 |
| RAM: | 64 Megabytes |
| Hard Disk: | 25 Megabytes of free storage space, plus room for data files |
| Display: | 800 x 600 resolution, 256 colors |

Recommended:

While Haestad Methods' software will perform adequately given the minimum system requirements, performance will only improve with a faster system. Our products are designed to perform at optimal levels with a fast CPU and ample amounts of RAM and free disk space. We highly recommend running our software on the best system possible to maximize its potential. We understand that an engineer's time is a valuable commodity, and we have designed our software to help make the most of that time.

### 1.4.2 Installing Haestad Methods' Products

For Windows 95, Windows 98, Windows NT 4.0, Windows 2000, Windows Me, and Windows XP follow these six easy steps, for installing a single user license copy of the program:

1. If you have not done so, turn on your computer.
2. Place the diskette labeled Disk 1 in the floppy disk drive (commonly the a: or b: drive).
3. Place the CD in your CD-ROM drive (commonly the d: or e: drive).
4. If the Autorun feature of the operating system is enabled, setup will begin automatically. Proceed to step six.
5. If Autorun is disabled, click the Start button on the task bar, select Run from the menu, and type d:\setup (use the actual drive letter of the CD-ROM drive if it is not the d: drive), and then click OK.
6. Follow the instructions of the Setup Wizard.

**If you own a network license version of the software, please refer to the Network Licensing section. If you still have questions, consult the KnowledgeBase on our web site www.haestad.com or contact Haestad Methods technical support.**

### 1.4.3 Uninstalling Haestad Methods' Products

Haestad Methods' products come with an uninstall option. After a single user license copy of a Haestad Methods' product is installed onto a computer, it must be uninstalled before a new installation can occur.

To uninstall the program:

On the Windows Start Bar, click **Start** / **Programs / Haestad Methods /** ***Product Name*** / **Uninstall** ***Product Name***. The original floppy disk labeled **Disk 1** that came with the product must be in the floppy drive at the time you uninstall.

### 1.4.4 Troubleshooting Setup or Uninstall

Because of the multi-tasking capabilities of Windows 95, 98, NT, 2000, Me, and XP you may have applications running in the background that make it difficult for the setup routines to determine the configuration of your current system. If you have difficulties during the install (setup) or uninstall process, please try these steps before contacting our technical support staff:

1. Restart your machine.
2. Verify that there are no other programs running. You can see applications currently in use by pressing Ctrl-Alt-Del in Windows 95, 98, Me, XP, or 2000, or Ctrl-Shift-Esc in Windows NT. Exit any applications that are running.
3. Run setup or uninstall again without running any other program first.

If these steps fail to successfully install or uninstall the product, contact our support staff immediately.

### 1.4.5 Software Registration

During the installation of the program, a dialog will appear asking you to register the software. Please note the label with your registration information is on the inside back cover of the manual.

Although this software is not copy protected, registration is required to unlock the software capabilities for the hydraulic features that you have licensed. All registration information must be entered into the **Registration** dialog exactly as it appears on the label.

- Company
- City
- State/Country
- Product ID
- Registration Number

After you have registered the software, you can check your current registration status by opening the registration dialog in the software itself.

To open the **Registration** dialog:

1. Select **Help / About** from the pull-down menus.
2. Click the Registration button in the **About** dialog.

The current registration status (number of licenses, expiration date, feature level, etc) will be displayed.

Use the **Print** button to print a copy of the information shown in the **Registration Form** dialog.

Use the **Copy** button to place the registration information in the Windows Clipboard so that you can paste it into another Windows application.

**If you own a network license version of the software, please refer to the Network Licensing section. If you still have questions, consult the KnowledgeBase on our web site www.haestad.com or contact Haestad Methods technical support.**

### 1.4.6 Upgrades

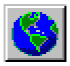
When you click the **Registration** button on the **Help / About *Product Name* …** dialog, the current registration status (number of licenses, expiration date, feature level, etc) is displayed. To upgrade to more pipes or inlets, higher feature levels, or additional licenses, contact our sales team today and request information on our ClientCare Program. We will provide the information you need to get up and running in no time!

### 1.4.7 Globe Button

Haestad Methods makes it easy to stay up-to-date with the latest advances in our software. Software maintenance releases can be downloaded from the Haestad Methods web site quickly and easily if you are a subscriber to our ClientCare Program. Just click the **Globe** icon on the tool palette to launch your preferred web browser and open the Haestad Methods' Program Update web site. The web site will automatically check to see if your installed version is the latest available. If it is not, it will provide you with the opportunity to download the correct upgrade to bring it up-to-date.

The ClientCare Program also gives you access to our extensive KnowledgeBase for answers to Frequently Asked Questions (FAQ). Contact our sales team for more information on our ClientCare Program.

**Use the Globe button to keep your investment current.**



## 1.4.8 Network Licensing

Network versions of this product are available. If you purchased a network version, your program will run at any workstation located on your network if a floating license key is available for use. Floating licenses allow one or more concurrent users of a particular application to access and use the full capabilities of the software if the number of concurrent licenses does not exceed the number allowed under the terms of the license sale. Once the number of concurrent users exceeds the licensed number, new application sessions will run in a limited demo mode.

Network licensing is implemented using Rainbow Industries SentinelLM™ license manager. Administrators should refer to the *SentinelLM™ System Administrators Guide* for details on implementing network licensing at your location.

### Registering Network Programs

During the installation of the network deployment folder, a dialog will appear asking you to register the software. The label with your registration information is on the inside back cover of the manual. This registration data is required to enable the software capabilities for the hydraulic network size and features that you have licensed. All registration information must be entered into the **Registration** dialog exactly as it appears on the label.

- Company
- City
- State/Country
- Product ID
- Registration Number

After you have registered the software, you can view the current registration and floating license usage status at any of the workstations that has the product software installed on it.

To open the **Registration** dialog:

1. Select **Help / About…** from the pull-down menus.
2. Click the **Registration** button.

The current registration status (number of floating licenses, expiration date, feature level, etc) will be displayed. If all available floating licenses are in current use, the software will run in demo mode.

Network administrators may activate network licenses and upgrade the features served by their floating licenses by invoking the **Request License** option, which is activated using the **Registration** button on this dialog.

Use the **Print** button to print a copy of the information shown.

Use the **Copy** button to export the registration information to the Windows Clipboard so that you can paste it into other Windows applications.

## Requesting Permanent Network License

System administrators who are responsible for managing network license versions of Haestad Methods' software must activate their organization’s floating licenses by obtaining a permanent license file from Haestad Methods. This may be done using the **Request License** button on the **Registration** dialog. This button will only be available for users who have purchased the network-licensing feature.

> **Haestad Methods uses SentinelLM License Manager software from Rainbow Technologies to manage network licensing for this application. For more information concerning the administration of the Haestad Methods floating network licensing, please refer to the Sentinel Software System Administrator’s Guide online documentation installed with your network license server software.**

To acquire a network license file, the administrator must first generate the network locking codes for the computer that will be acting as the network license server. To get you license server locking code, use the SentinelLM ***echoid*** utility. This is installed with the license server software onto the computer acting as the network license host for this application.

> **The echoid utility must be run from the same computer that will act as the license server host for this particular Haestad Methods application.**

Write down the values for the locking codes that are posted in the echoid utility’s message box. Be certain to record these values accurately, as they will be used by Haestad Methods to generate a custom license file keyed to the specific license server’s hardware signature. Once issued, a license key-code may not be installed on another machine. You will not be able to transport the license server to another network machine without obtaining new lock codes.

With echoid values in hand, start the Haestad Methods product application on any workstation located on the network served by the license manager. You can even install and run the Haestad Methods application from the same computer that will be acting as the license server host computer.

Select **Help / About…** from the pull-down menus to open the **Registration** dialog. Open the **Request License Key** dialog using the **Request License** button. Fill out the form, then either email or fax the completed form to Haestad Methods using the **Submit Request** button.

## Installation Guide for Network License Versions

To set up a Haestad Methods’ software product for operation as a network licensed version:

1. Place the diskette labeled Disk 1 in the floppy disk drive (commonly the a: or b: drive).
2. Place the CD in your CD-ROM drive (commonly the d: or e: drive).
3. If the Autorun feature of the operating system is enabled, setup will begin automatically. Proceed to step 5.
4. If Autorun is disabled, click the **Start** button on the task bar, select **Run** from the menu, and type d:\setup (use the actual drive letter of the CD-ROM drive if it is not the d: drive). Click **OK**.
5. To perform the following steps, you must have full administrator privileges for the target network-based installation folders. Follow the instructions of the **Setup Wizard**, which will guide you through the installation of two components:
   – **Network deployment folder** - A directory installed on a network node that is available from all client workstations on which the license product will be installed. Users of the floating licenses will invoke the network-based installation utility ***setup.exe***, which will install and configure the application to each client workstation.

- **Network License Manager and Utilities** - The license manager service executable file that will automatically monitor availability and distribute network floating licenses to client applications as they are started up across the license hosting LAN. The license manager may be installed on any shared node in the network, but is generally located on a network server machine.

6. Start the license server using the appropriate procedure for the host machine's operating system:
   - **Windows NT/2000** - Use the **loadls** utility via the **Service Loader** menu option to install the license server. The license manager runs as a service and can be manually controlled via the **Windows NT / 2000 Control Panel / Services** group.
   - **Windows 95/98/ME** - To enable automatic license server startup, use **Windows Explorer** to add the license server program, Lserv9x.exe, to the Windows 95/98 system **StartUp** folder. Manually start the license server by running the **lserv9x** via the **License Server** menu option.

7. Announce the availability of the product via email. Instruct interested users to install the product the using the **Start / Run…** menu command and browsing to the network deployment folder installed in step 5 to run **setup.exe**. The license server ships with special 30-day licenses that will allow users to begin using the application immediately.

8. Obtain a permanent license file for the application. A permanent license file must be obtained from Haestad Methods within 30 days of receipt of the product package. Request a permanent license file by following these steps:
   - At the host computer on which the license server will run, use the **echoid** utility via the **Locking Codes** menu option to determine the locking codes that will be used to generate license keys for your network. The license key file will be configured specifically for the license server machine installation. Write these locking codes down.
   - Start the Haestad Methods' product application at any workstation where the product has been installed. The product can even be installed directly on the computer acting as the license server.
   - Select **Help / About …** from the pull-down menus and click the **Registration** button. Use the **Request License** button at the bottom of the **Registration** dialog and fill out the License Key Request form with the system administrator contact and host server locking code information. Be certain to accurately record the locking codes obtained during the previous step since inaccurate information will result in the generation of unusable license files.
   - Click the **Submit Request…** button. Following the instructions in the form, e-mail or fax the form to Haestad Methods. The activation request will be processed, and a license file will be generated and e-mailed to the system administrator making the request.

9. Use the ***lslic*** utility located in the **AdminTools** directory to modify the permanent license file managed by the network license server. After the license key file requested above is received via e-mail from Haestad Methods, save the file attachment to a computer folder on any computer resident on the network serviced by the running license server. For future convenience, safety, and ease-of-support, it is recommended that the license file be saved in the license manager tools directory, **AdminTools**. This utility must be run from the operating system prompt. Enter **`lslic -F <filename>`**, where `<filename>` is the name of the license file attachment emailed by Haestad Methods and saved to the hard-drive. This step will install the new license key into the license file, **lservrc**, located on the same computer and in the same directory that the license server resides.

Once these steps are completed, floating licenses will be available to concurrent users via the network. Should the number of users exceed the number of license keys available, the unlicensed client sessions will continue to run in demo mode.

### Network Deployment Folder

Interested users may install the complete product via the network deployment folder using the Windows **Start / Run** command. Browse to the deployment directory, and run **setup**.**exe** to install the program to a client workstation.

# 1.5 Learning SewerCAD

## 1.5.1 SewerCAD Documentation

SewerCAD's on-line documentation delivers extensive detail and convenient assistance. Simply click the Help button, press the **F1** key, or right-click anywhere in the program to access context-sensitive help.

The SewerCAD User's Guide is provided to you as a means to read and learn about SewerCAD while you are away from your computer. The topics you find in the User's Guide will also be found in the on-line help.

SewerCAD also contains on-line tutorials, lessons, and sample files to get you familiar with the features and capabilities of SewerCAD. The tutorials can be accessed by clicking **Help / Tutorials** from the pull-down menus. The lessons can be found in the printed documentation, as well as in the on-line Help. The sample files are located in your Haestad\SWRC\Sample directory.

## 1.5.2 How to Use Help

All of our products feature extensive context-sensitive help. There are several ways to obtain help on topics:

- Select **Help** from the pull-down menu.
- To get help for the window in which you are working, press the **F1** key or click the **Help** button.
- To get help for a specific item, right-click the desired item and select **Help** from the pop-up menu.

**To navigate within Help:**

- When you click text that is underlined, Help "jumps" to the related topic or definition. If the text is dashed underlined, the text will appear in a pop-up window.
- To return to the previous topic, click the **Back** button at the top of the **Help** window.
- To print a Help topic, click the **Print** button at the top of the **Help** window.

**To make the Help window stay on top of other windows, select Options / Keep On Top / On Top from the pull-down menu in the main Help window.**

## 1.5.3 How Do I?

**How Do I** is an easily referenced topic in SewerCAD's on-line documentation. It is a listing of commonly asked questions about SewerCAD. Just follow these steps to find your way to How Do I**:**

1. Click **Help / How Do I** from the pull-down menu
2. The listing of **How Do I** topics will appear. Just click the topic of your choice for a detailed explanation.
3. To return to the listing of **How Do I** questions, click the **Back** button.

### Help

Provides context sensitive help for the current window.


## 1.5.4 Glossary

The glossary contains many terms used throughout the application and the on-line Help.

To use the Glossary:

1. Click **Help / Contents** to open the main **Help** window.
2. Click the **Contents** tab, and then double-click the **Glossary** book.
3. Click the **Glossary** page, and the Glossary topic will appear.
4. Click the first letter of the word for which you are looking for more information.
5. Click the desired term, and a pop-up box will appear with a definition of the selected word.

## 1.5.5 Tutorials

Tutorials provide a quick introduction to various features of the program. To access tutorials, click **Help / Tutorials** from the pull-down menus. Run a tutorial by selecting one of the entries in the list and clicking the **OK** button.

End a tutorial at any time by either pressing the **Esc** key (in Stand-Alone mode), or by clicking the **Close** button in the upper right-hand corner of any tutorial dialog. If you need further information, access our on-line help by pressing the **F1** key.

## 1.5.6 Sample Projects

To explore one of the sample projects provided to demonstrate this software's capabilities:

1. Select **File / Open** from the pull-down menus to access the Open Project File dialog.
2. Choose Sample.swr (or Sample.dwg, if using AutoCAD mode) from the Sample directory, and click Open.

These are working network models, so you can explore the system and see how different elements are modeled. First, calculate the system by using the **GO** button on the main toolbar. Then use Quick View, Graphs, Profiles, Tabular views, Detailed Reports, and Color Coding to see how the system behaves. To get the best introduction to a new feature, try running some of the tutorials.

## 1.5.7 Haestad Methods Workshops and Certification

Haestad Methods offers a variety of workshops dealing with topics ranging from urban stormwater management to water distribution modeling, alternating theory, modeling insights and hands-on practice with the software. These workshops are held at various locations throughout the world, and include discounted pricing when purchasing Haestad Methods software.

For more information on our workshops (such as instructors, schedules, pricing, and locations), please contact our sales department, or visit our web site at www.haestad.com for current workshop schedules and locations. We will be glad to answer any questions you may have regarding the workshops and our other products and services.

# 1.6 Contacting Haestad Methods

## 1.6.1 Sales

Haestad Methods' professional staff is ready to answer your questions. Please contact your sales representative for any questions regarding Haestad Methods' latest products and prices.

Phone: +1-203-755-1666

Fax: +1-203-597-1488

e-mail: Sales@haestad.com

## 1.6.2 Technical Support

We hope that everything runs smoothly and you never have a need for our technical support staff. However, if you do need support our highly skilled staff offers their services **seven days a week**, and may be contacted by phone, fax, and the Internet. For information on the various levels of support that we offer, contact our sales team today and request information on our ClientCare Program.

**When calling for support, in order to assist our technicians in troubleshooting your problem, please be in front of your computer and have the following information available:**

- Operating system your computer is running (Windows 95, Windows 98, Windows NT, Windows Me, Windows 2000, or Windows XP)
- Name and version number of the Haestad Methods software you are calling about
- Version of AutoCAD you are running (if applicable)
- Any error messages or other information that was generated
- A note of exactly what you were doing when you encountered the problem

**When e-mailing or faxing for support, please provide the following additional details to enable us to provide a timely and accurate response:**

- Company name, address, and phone number
- A detailed explanation of your concerns
- The Haestad.log and Error.log files located in the product directory (e.g. Haestad\SewerCAD).

Hours:

Monday - Friday: 9:00 AM to 8:00 PM EST

Saturday and Sunday: 9:00 AM to 5:00 PM EST

**You can contact our technical support team at:**

Phone: +1-203-755-1666

Fax: +1-203-597-1488

e-mail: Support@haestad.com

### 1.6.3 Your Suggestions Count

At Haestad Methods, we strive to continually provide you with sophisticated software and documentation. We are very interested in hearing your suggestions for improving our products, our on-line Help system, and our printed manuals. Your feedback will guide us in developing products that will make you more productive.

Please let us hear from you!

### 1.6.4 How to Contact Us

Phone: +1-203-755-1666

Fax: +1-203-597-1488

e-mail: Support@haestad.com

Info@haestad.com

Sales@haestad.com

Internet: http://www.haestad.com

Mail: Haestad Methods

37 Brookside Road

Waterbury, CT 06708-1499

USA

# Chapter 2

# SewerCAD Main Window

## 2.1 Overview

If you are already familiar with standard Windows interfaces, you will find SewerCAD to be intuitive and comfortable. Even if you are not accustomed to Windows standards, just a few minutes of exploring SewerCAD should be enough to acquaint you with the flexibility and power that this program offers.

In this chapter, we will examine the program's main window, menus, and toolbars. Additional tools for layout, annotating and editing are described in the chapter "Layout and Editing Tools." After reading this chapter, you should be able to interact with SewerCAD in a quick and efficient manner.

## 2.2 Main Window Components

### 2.2.1 Stand-Alone Mode, AutoCAD Mode

Both the Stand-Alone graphical editor and the AutoCAD interface perform modeling activity through the SewerCAD model server.

This use of a common central model enables both modes to perform the same functions with the same behaviors, so things like graphical layout and model management are virtually identical between the two modes.

One advantage of Stand-Alone mode is that your interaction is more streamlined and dynamic by virtue of the fact that the editing environment is a dedicated network editor. Also, since AutoCAD is not needed to run in Stand-Alone mode, less system resources and memory are used.

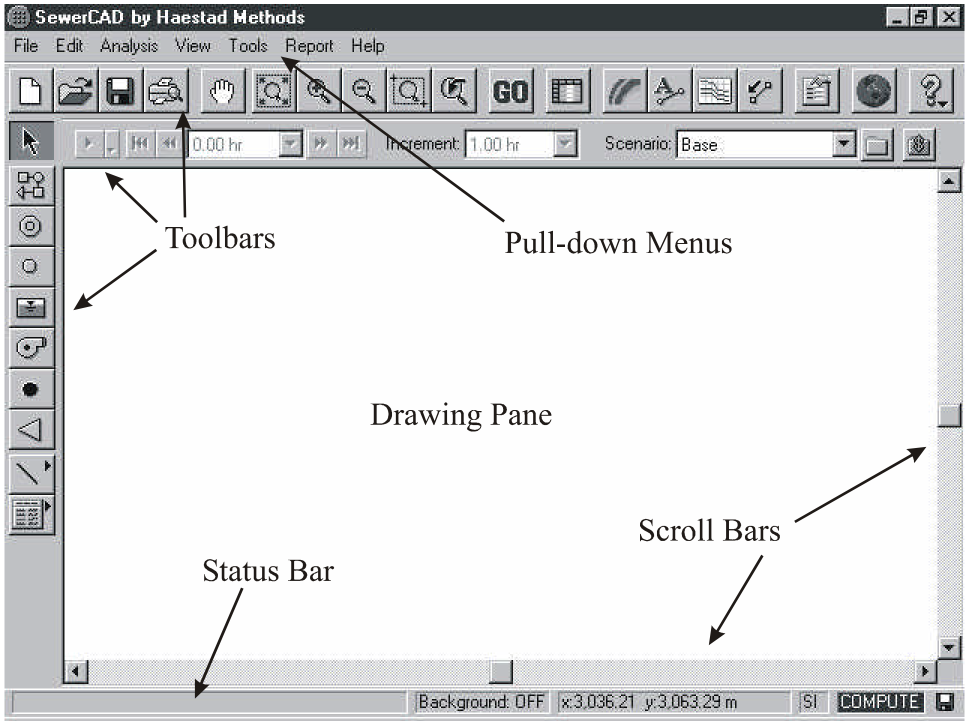
A significant advantage of the AutoCAD mode is that you can create and model your network directly within your primary drafting environment. This gives you access to all of AutoCAD’s powerful drafting and presentation tools, while still enabling you to perform SewerCAD modeling tasks like editing, solving, and data management. This relationship between SewerCAD and AutoCAD enables extremely detailed and accurate mapping of model features, and provides the full array of output and presentation features available in AutoCAD. This facility provides the most flexibility and the highest degree of compatibility with other CAD-based applications and drawing data maintained at your organization.

**AutoCAD mode is an available feature level. Contact us to upgrade your SewerCAD Stand-Alone version to include the AutoCAD mode feature level.**

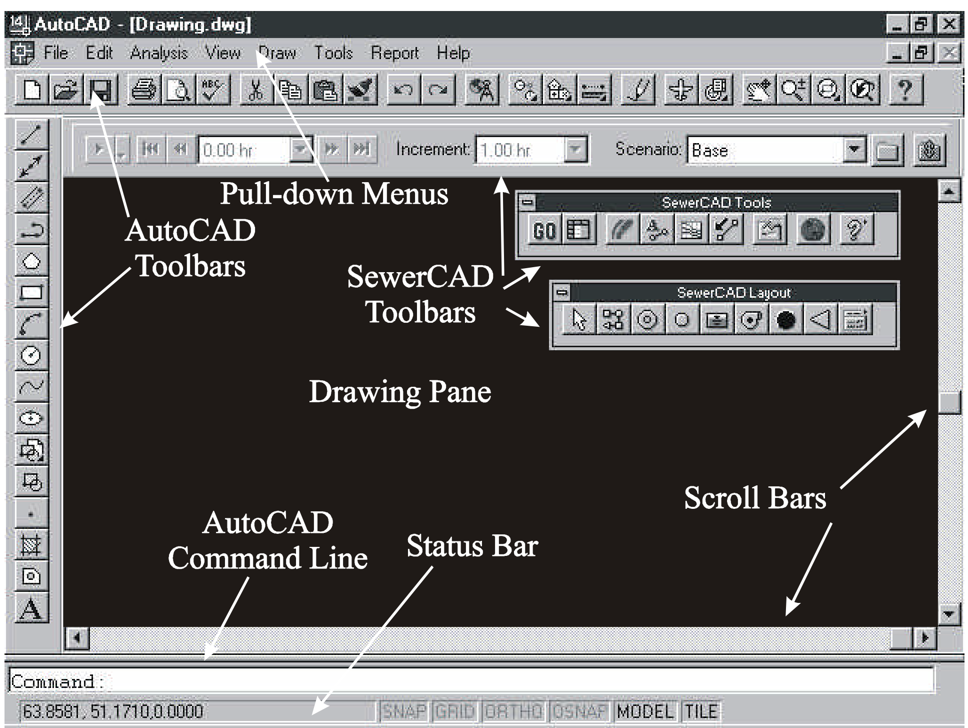
### 2.2.2 SewerCAD Main Windows

Both the SewerCAD Stand-Alone interface and the AutoCAD interface have many components common to Windows based programs. The following figures illustrate some of the important areas that make up the SewerCAD Stand-Alone, AutoCAD R14 and AutoCAD 2000 windows, respectively.

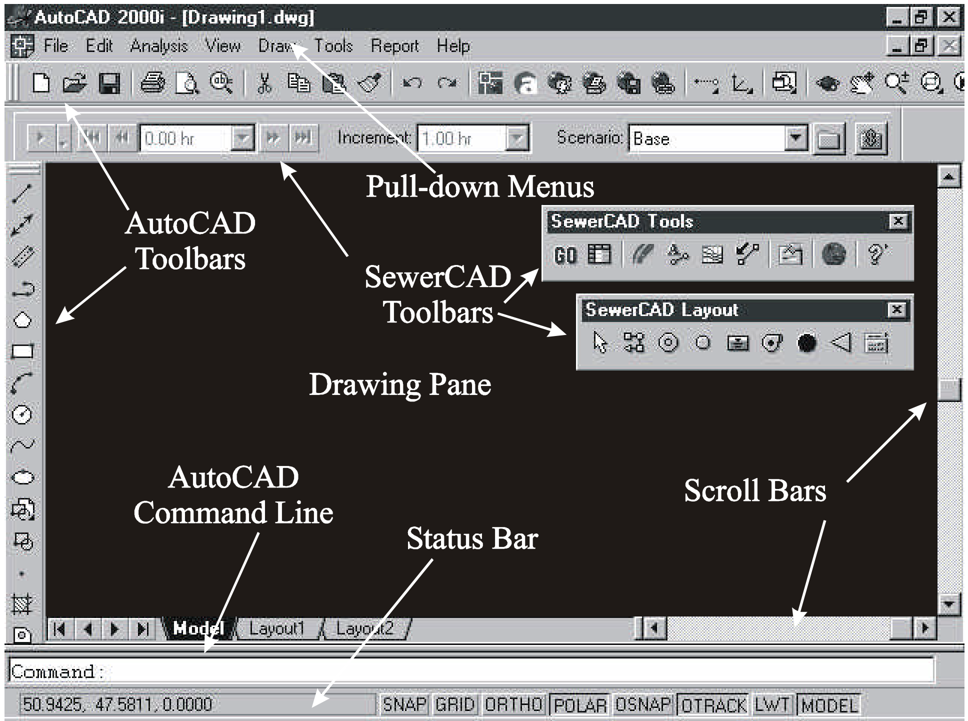
SewerCAD Stand-Alone Interface




SewerCAD AutoCAD R14 Interface



SewerCAD AutoCAD 2000 Interface




Notice that many of the window components, such as menus and toolbars, are very similar for the Stand-Alone editor and AutoCAD. Other features, such as the command line, are only available in AutoCAD.

For more information regarding the various functions and behaviors of AutoCAD, please refer to your AutoCAD documentation.

## 2.2.3 Drawing Pane

The drawing pane is the center of SewerCAD's graphical activity, where the sanitary sewer network elements are displayed. It is the main interactive area for creating elements, editing data, and even displaying results.

In Stand-Alone mode, the drawing pane can also display a single .DXF background image. This background can be helpful for aligning and positioning elements, as well as adding additional drafting elements for plotting plan views.

In AutoCAD, the drawing pane is where all graphical elements, not just SewerCAD entities, are displayed and manipulated. Lines, arcs, text, and many other drafting elements can be created and modified within the drawing pane.

## 2.2.4 Status Pane

The status pane is located along the bottom of the main application window and provides useful information about application settings, the current user activity, file save status, etc.

**When you position the mouse pointer over a toolbar button or menu item, the status pane will display a descriptive message. Leave the mouse over a section of the status pane to display an informative popup tip.**

## 2.2.5 Menus, Toolbars, and Shortcut Keys

Anyone who has ever watched someone else use a computer should realize that not all people use computers in the same way. Some prefer to primarily use the mouse, some the keyboard, and others use a mixture of both.

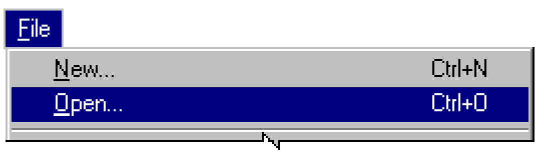
For this reason, Haestad Methods' programs provide access to the most common features through several means, including:

- Pull-Down Menus
- Toolbars
- Shortcut Keys
- Command Line (AutoCAD Only)

### Pull-down Menus

As with any Windows-based program, the menu system provides easy access to many features. Items can be accessed by clicking the desired menu text, or by pressing the **Alt** key to activate the menus and then pressing the key for the underlined letter of the menu item you wish to access.

For example, to open an existing file you can use the mouse to select **File / Open,** or you can press the **Alt** and **F** keys (**Alt** + **F**), then **O** on the keyboard.




### Toolbars

Toolbar buttons offer one-click access to some of the most commonly used features, giving you a quicker way to perform the most frequent operations.

For example, to open an existing file (the equivalent of selecting **File / Open** from the pull-down menu), simply click the **File Open** tool.

### Shortcut Keys

Shortcut keys are the keyboard equivalent of toolbars. Key combinations - usually a simultaneous application of the **Ctrl** (Control) key and a letter key - can provide instant access to common features. If a shortcut is available for a menu item, it will be indicated in the menu itself.

For example, to open an existing file (the equivalent of selecting **File / Open** from the pull-down menus) you can press the **Ctrl** and **O** keys (**Ctrl + O**) at the same time.

## 2.2.6 Command Line

The command line is a special area that is not available in Stand-Alone mode. In AutoCAD mode, this area enables you to type commands directly, rather than using the menus, toolbars, or shortcut keys.

For example, to open an existing AutoCAD file (the equivalent of selecting **File / Open** from the pull-down menus) you can simply type the command **OPEN** at the AutoCAD command line.

Many of AutoCAD's commands are easy to enter at the command line, including accessing drafting tools (like **LINE** and **CIRCLE**) and editing tools (like **MOVE** and **ERASE**). Modeling elements can also be manipulated through the AutoCAD command line, just as they can be manipulated via the menus or toolbars.

For more information on the AutoCAD command line, please see the AutoCAD documentation.

# 2.3 SewerCAD Menus

## 2.3.1 Pull-down Menus

Although the toolbars and shortcut keys provide quick and easy access to commonly used features, the pull-down menu system provides much more comprehensive access to SewerCAD's properties and behaviors. Since toolbar buttons and shortcut keys do not exist for all of these features, the menus are a logical choice for exploring all areas of SewerCAD. This section will introduce you to many of SewerCAD's features and show you how you can access these features, including any toolbar buttons and shortcut keys that are available.

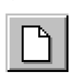
Commands are grouped under several menus, which consist of the following selections:

- File
- Edit
- Analysis
- View
- Tools
- Report
- Draw (in AutoCAD mode only)
- Help

## 2.3.2 File Menu

The **File** menu contains many of the items dealing with project management. It provides features to create, read, write, and print project files, as well as features for sharing data with databases and geographical information systems.

- **New** - Create a new project. When you choose this item, a dialog will appear so that you can enter a drive, directory, and filename for your new project file. The **Project Setup Wizard** will then help you set up your new project.

  Toolbar Button: 

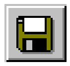
  Shortcut Key: **Ctrl + N**

- **Open** - Load an existing project file from disk. When you select this item, a dialog will appear so you can choose the name and location of the project you want to open.

  Toolbar Button: 

  Shortcut Key: **Ctrl + O**

- **Save** - Save the current project file to disk. While saving the project file, the status pane will display a message to show you the progress of the save command.

  Toolbar Button: 

  Shortcut Key: **Ctrl + S**

- **Save As** - Save the current project to disk under a different filename. When you use this command, a dialog will appear prompting you to enter the drive, directory, and a new file name for your project.
- **Project Summary** - Access the Project Summary information, such as the project title and the project engineer.
- **Import / Database** - Import data from a Microsoft Access database (.mdb) using one of the default database import connections included with the product.
- **Import / Shapefile** - Build network elements from ESRI Shapefiles. This command will start the **Shapefile Wizard**, which will help you bring the GIS elements and their associated data into your project.
- **Import / Polyline to Pipe** (Stand-Alone Mode Only) - Build network elements from a .DXF file. This command will start the **Polyline to Pipe Wizard**, which will help you convert polyline data representing geographical data into your project as pipes and nodes. A similar command called ‘Change Entities to Pipes’ is available in the AutoCAD version.
- **Import / Land Development Desktop** - Build SewerCAD network elements from Land Development Desktop Pipes data. This command will start the **Land Development Desktop Import Wizard**, which will help you bring the data contained in the Pipes module into your SewerCAD project.
- **Import / Loading Data** - Import hydrograph and temporal pattern data from an ASCII text file.
- **Import / DXF Background** (Stand-Alone Mode Only) - Bring in a single .DXF drawing file into your project as a background map. This command will open a dialog that prompts you to select the name and location of the desired .DXF file.
- **Import / SewerCAD** (AutoCAD Mode Only) - Import a SewerCAD (.swr) file into SewerCAD in AutoCAD mode.
- **Export / Database** - Export data to a new Microsoft Access database (.mdb) using one of the default database export connections included with the product.
- **Export / Shapefile** - Export your project to ESRI Shapefiles for use with GIS applications. This command will start the **Shapefile Wizard**, which will help you create Shapefiles with the desired project elements and their associated data.
- **Export / Land Development Desktop** - Export SewerCAD network elements into a Land Development Desktop project. This command will start the **Land Development Desktop Export Wizard**, which will help you bring the data contained in your SewerCAD project into the Land Development Desktop through the Pipes module.
- **Export / DXF File** - Export the entire network drawing to a .DXF file, which can be read by all popular CAD programs. This command will open a dialog prompting you to enter the name and location of the .DXF file you would like to create.
- **Synchronize / Database Connections** - Access the **Database Connection Manager**, which allows you to share SewerCAD data with external databases, spreadsheets, and other ODBC compliant sources. Details of this comprehensive feature are explained in the chapter entitled "GIS and Database Connections."

- **Synchronize / Shapefile Connections** - Access the **Shapefile Connection Manager**, which allows you to share SewerCAD data with external GIS projects. Details of this feature are explained in the chapter entitled "GIS and Database Connections."
- **Print** - Print the current view of the project drawing to a printer. Profiles and tabular reports are printed from their respective windows. The print command invokes the standard **Print** dialog, which allows you to select things such as the printer to be used, printer properties, and the print range.

  Shortcut Key: **Ctrl + P**

- **Print Preview** - Open the **Print Preview** dialog for the current view of the project drawing. This feature allows you to see the drawing as it will be printed before sending it to the printer.

  Toolbar Button: 

- **Print Setup** - Select the default printer for SewerCAD to use. You can also use this command to change options related to the printer driver, such as resolution and portrait or landscape orientation.
- **Exit SewerCAD** - Close the current project and exit SewerCAD. If you made any changes to the current project, you will be asked if you want to save the project before you exit SewerCAD.

  Shortcut Key: **Alt + F4**

- **1, 2, etc.** - The most recently opened project files appear at the bottom of the File menu. Using this file list, you can quickly select and open a recently used file without navigating to its drive and directory.

### 2.3.3 Edit Menu

The **Edit** menu provides access to basic commands for controlling SewerCAD elements, such as element navigation, selection, deletion, and undo and redo.

- **Undo [Last Action Performed]** - Reverse the last reversible action performed. Reversible include element creation, element deletion, element editing, and moves. The effects of some model actions cannot be reversed, such as calculation, database synchronization, scenario creation and tabular edits. Additionally, to ensure that the model is maintained in a consistent state, the undo/redo history is flushed whenever an irreversible menu or button command is issued.

  Shortcut Key: **Ctrl + Z** (not available in AutoCAD Mode)

- **Redo [Last Action Undone]** - Reverse the effects of the last undo action. Any action that can be undone can be redone.

  Shortcut Key: **Ctrl + Y** (not available in AutoCAD Mode)

- **Cut** (AutoCAD Mode Only) - Delete the selected entities and place them on the Windows clipboard. These items can be pasted into other Window programs or back into AutoCAD.

  Shortcut Key: **Ctrl+X**

- **Copy** (AutoCAD Mode Only) - Place a copy of the selected entities from the current AutoCAD drawing to the Windows clipboard.

  Shortcut Key: **Ctrl+C**

- **Paste** (AutoCAD Mode Only) - Place the items on the Windows clipboard into AutoCAD.

  Shortcut Key: **Ctrl+V**

- **Paste Special** (AutoCAD Mode Only) - Place special items on the Windows clipboard, such as Excel Spreadsheets and Word documents, into AutoCAD.

- **Delete** (Stand-Alone Mode Only) - Erase selected elements. Deleting an element removes it from all aspects of the project, including all scenarios.

  Shortcut Key: **Delete**

- **Select / All** - Select all of the elements in the current project.

  Shortcut Key: **Ctrl + A** (not available in AutoCAD Mode)

- **Select / By Element / All [Element Type]** - Select all elements of a certain type, such as all pipes or all junctions.
- **Select / By Selection Set** - Select the elements contained in a predefined selection set.
- **Select / Clear Selection** - Reset (empty) the current selection set.
- **Edit Element** (AutoCAD Mode Only) - Open the element's dialog. Select this item from the pull-down menu, and click the element you wish to edit.
- **Edit Elements** (AutoCAD Mode Only) - Edit a group of elements. Select this item from the pull-down menu, then select a group of elements using the crosshairs or by windowing an area. After the elements have been selected, the **Table Manager** dialog will appear.
- **Modify Elements / Change Pipe Width** (AutoCAD Mode Only) - Modify the width of the AutoCAD objects representing pipes in the current selection set.
- **Modify Elements / Scale Elements** (AutoCAD Mode Only) - Scale the symbols representing the elements in the current selection set.
- **Modify Elements / Rotate Elements** (AutoCAD Mode Only) - Rotate the labels of the elements in the current selection set.
- **Modify Pipes / Insert Bend** (AutoCAD Mode Only) - Insert a bend along a selected Pressure Pipe Element.
- **Modify Pipes / Remove Bend** (AutoCAD Mode Only) - Delete a bend along a selected Pressure Pipe Element.
- **Modify Pipes / Remove All Bends** (AutoCAD Mode Only) - Delete all the bends along a selected Pressure Pipe Element.
- **Modify Pipes / Change Widths** (AutoCAD Mode Only) - Change the Width of the lines representing pressure pipes.
- **Change Entities to Pipes** (AutoCAD Mode Only) - Build network elements from AutoCAD entities. This command will start the **Polyline to Pipe Wizard**, which will help you convert the desired polyline data representing geographical data into pipes. A similar command called 'Import Polyline to Pipe' is available in the Stand-Alone version under the **File** pull-down menu.
- **Find Element** - Open the **Find Element** dialog, which allows you to locate an element and bring it to the center of the drawing pane. This element search is based on the element label, and is not case sensitive.

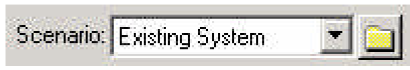
  Shortcut Key: **Ctrl + F** (not available in AutoCAD Mode)

- **Review Drawing** - Open the **Drawing Review** window, which allows you to isolate elements that may need to be scrutinized for potential problems, such as orphaned nodes, elements with messages, or nodes in close proximity to each other.

### 2.3.4 Analysis Menu

The **Analysis** menu holds items regarding calculations. These include such items as scenario access and calculation command.

- **Scenarios** - Opens the **Scenario Manager**, where you can create, compare, and otherwise manipulate scenarios.

  Toolbar Button: 


- **Alternatives** - Provides access to the **Alternative Manager**, where you can add, delete, and otherwise manipulate alternatives.
- **Patterns** - Opens the **Pattern Manager** where you can create and edit diurnal loading patterns for use with extended period simulations.
- **Extreme Flows** - Opens the **Extreme Flows Manager** where you can create a list of extreme flow setups. Flow setups can then be associated with a scenario to control the peaking of the base sanitary loads.
- **Pattern Setups** - Opens the **Pattern Setup Manager** where you can associate diurnal patterns with the appropriate unit sanitary loads for a given scenario.
- **Default Design Constraints** - Specify constraints for pipes, nodes, and inlets to be used during an automatic design and while checking constraints for an analysis calculation.
- **Compute Costs** - Opens the **Cost Manager** to view, edit, or perform Cost Estimating calculations.

  Toolbar Button: 

- **Compute** - Opens the **Calculation Dialog**. This dialog gives you access to items such as referenced alternatives and calculation options.

  Toolbar Button: 


### 2.3.5 View Menu

In both AutoCAD and Stand-Alone mode, the **View** menu provides access to tools dealing with the drawing pane, toolbar visibility, and so forth.

*In Stand-Alone mode, you are provided with the following tools:*

- **Pan -** Upon selection hold down the left mouse button to move the drawing

  Toolbar Button: 

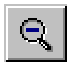
  Shortcut Key: Hold down the mouse wheel.

- **Zoom In** - Enlarge the current view of the drawing.

  Toolbar Button: 

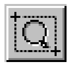
  Shortcut Key: + (Keypad)

- **Zoom Out** - Reduce the current view of the drawing.

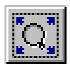
  Toolbar Button: 

  Shortcut Key: - (Keypad)

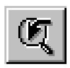
- **Zoom Window** - Activate the user-defined zoom tool. This tool enables you to select the corners of the area within the drawing pane that you wish to enlarge.

  Toolbar Button: 

- **Zoom Extents** - Reset the drawing zoom factor such that all elements are displayed in the drawing pane.

  Toolbar Button: 

- **Zoom Previous** - Return the drawing pane to the most recent view.

  Toolbar Button: 

- **Zoom Center** - Open the **Zoom Center** dialog, which enables you to specify the central coordinates and zoom factor to change the view in the drawing pane.
- **Aerial View** - Enable or disable the **Aerial View** window. This window allows you to display a second view of the drawing at a larger scale.
- **Quick View** - Enable or disable the **Quick View** window, which allows you to quickly view input and output data for any element.

  Toolbar Button: 

- **Toolbars / Standard** - Toggle the display of the **Standard** toolbar at the top of the window, which provides shortcuts to the most commonly used commands.
- **Toolbars / Analysis Toolbar** - Toggle the display of the **Analysis** toolbar, which includes the scenario selection list, as well as the time step selection list if applicable.
- **Status Pane** - Toggle the display of information at the bottom of the window regarding your current project.
- **Background** - Toggle the visibility of the project's .DXF background. If there is no .DXF background specified for the current project, this menu item will be disabled.

**In AutoCAD mode, refer to the AutoCAD on-line help.**

### 2.3.6 Tools Menu

The **Tools** menu allows you access to many useful features for viewing results, as well as selecting the tools used to generate network elements and graphical annotations within the drawing pane.

- **Selection Sets** - Opens the **Selection Set Manager**, which allows you to predefine a group of network elements that you want to manipulate together.

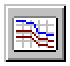
- **Color Coding** - Opens the **Color Coding** dialog.  Color coding can be used to control the geographical display of elements based on value ranges such as pipe diameter and hydraulic grade.

  Toolbar Button: 

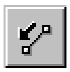
- **Element Annotation** - Provides access to the **Element Annotation** dialog, which allows you to display additional element attribute labeling such as pipe diameter and outlet total flow.

  Toolbar Button: 

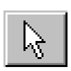
- **Profiling** - Opens the **Profiles** dialog where you can create new profiles or view previously created ones.

  Toolbar Button: 

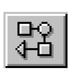
- **Diversion Network** - Displays a view of the **Diversion Network**, which is implicitly defined from the diversion targets.

  Toolbar Button: 

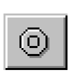
- **Relabel Elements** - Open the **Relabel Elements** dialog, which enables you to renumber some or all of your project elements.
- **Element Labeling** - Set the format for the labels applied to elements as they are added to the drawing.
- **Prototypes** - Specify the default values for new network elements.
- **Engineering Libraries** - Declare the paths to and edit the libraries used in this project.
- **User Data Extension** - Open the **User Data Extension** dialog, where you can add and define custom data fields.  For instance, you can add new fields such as the pipe installation date.
- **FlexUnits** - Open the **FlexUnits** dialog, where you can control units and display precision for any parameter.  You can also change the unit and display precision of variables from several other areas within the program.
- **Layout / Select** - Activates the **Selection** tool, which is used to select elements within the drawing pane.  Once elements are selected, they can be edited, moved, and otherwise manipulated.

  Toolbar Button: 

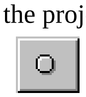
- **Layout / Pipe** - Activates the **Pipe Layout** tool, which enables you to connect nodes (new or existing) with a new pipe element.

  Toolbar Button: 

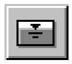
- **Layout / Manhole** - Activates the **Manhole** tool, which is used to add manhole elements to the project.

  Toolbar Button: 

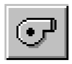
- **Layout / Junction Chamber** - Activates the **Junction Chamber** tool, which is used to add junction chamber elements to the project.

  Toolbar Button: 

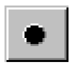
- **Layout / Wet Well** - Activates the **Wet Well** tool, which is used to add wet wells to the project.

  Toolbar Button: 

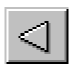
- **Layout / Pump** - Activates the **Pump** tool, which is used to add pumps to the project.

  Toolbar Button: 

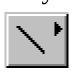
- **Layout / Pressure Junction** - Activates the **Pressure Junction** tool, which is used to add pressure junctions to the project.

  Toolbar Button: 

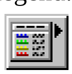
- **Layout / Outlet** - Activates the **Outlet** tool, which is used to add outlet elements to the project.

  Toolbar Button: 

- **Layout / Graphic Annotation** - Activates various annotation tools, which enable you to add lines, text elements, and other non-hydraulic elements to the project drawing.

  Toolbar Button: 

- **Layout / Legend** - Activates tools that enable you to add legends to the project drawing, such as a pipe or node color-coding legend.

  Toolbar Button: 

- **Options** - Activates the **Options** dialog, where you can customize the **Global**, **Project**, and **Drawing Options** for your projects.
- **Element Properties** (AutoCAD Mode only) - Opens a dialog that allows you to establish the layers and style of SewerCAD hydraulic elements and text.
- **Preferences** (AutoCAD Mode only) - Opens the AutoCAD Preferences dialog. See the AutoCAD on-line help for more information.

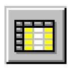
### 2.3.7 Report Menu

The **Report** menu provides access to a collection of preformatted textual and graphical reports. Furthermore, it provides access to FlexTables, which enable you to create your own custom reports.

- **Element Details** - Opens the **Detailed Reports** dialog, which enables you to print detailed reports for any set of elements.
- **Element Results** - Opens the **Analysis Results Report** dialog, which enables you to print reports of the results for any set of elements.
- **Element Hydrographs** - Opens the **Graph Setup** dialog from which you can generate hydrographs for multiple elements within the hydraulic network over multiple scenarios.
- **Element Graphs / Gravity Pipes** - Opens the **Graph Setup** dialog from which you can graph various attributes associated with gravity pipes over time.
- **Element Graphs / Pressure Pipes** - Opens the **Graph Setup** dialog from which you can graph various attributes associated with pressure pipes over time.
- **Element Graphs / Manholes** - Opens the **Graph Setup** dialog from which you can graph various

attributes associated with manholes over time.

- **Element Graphs / Junction Chambers** - Opens the **Graph Setup** dialog from which you can graph various attributes associated with junction chambers over time.
- **Element Graphs / Wet Wells** - Opens the **Graph Setup** dialog from which you can graph various attributes associated with wet wells over time.
- **Element Graphs / Pumps** - Opens the **Graph Setup** dialog from which you can graph various attributes associated with pumps over time.
- **Element Graphs / Pressure Junctions** - Opens the **Graph Setup** dialog from which you can graph various attributes associated with pressure junctions over time.
- **Element Graphs / Outlets** - Opens the **Graph Setup** dialog from which you can graph various attributes associated with outlets over time.
- **Tables** - Provides access to the **Table Manager**, which enables you to open predefined tables or generate custom tables.

  Toolbar Button: 

- **Scenario Summary** - Generates a report for the current scenario, including things such as referenced alternatives and calculation options.
- **Project Inventory** - Generates a report summarizing the project elements, including the number and breakdown of pipes and the number of manholes.
- **Plan View** - Generates plan view printable reports of the network, for either the current drawing display (Current View) or the entire drawing extents (Full View).

### 2.3.8 Draw Menu (in AutoCAD Mode Only)

The **Draw** menu is actually an AutoCAD menu that is accessible in the current program.

> **You can add additional AutoCAD menus to your Haestad Methods' application menus using AutoCAD's 'menuload' command. See the AutoCAD documentation for more information.**

### 2.3.9 Help Menu

The **Help** menu contains items that relate to on-line documentation for SewerCAD. Help includes the information contained in the printed documentation, as well as updated information and interactive tutorials.

**Help** menu items can also be accessed from the **Help** button:

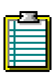
- **Contents** - Opens the table of contents for the on-line help.
- **How to Use Help** - Provides access to instructions for using the help system.
- **Release Notes** - Provides the latest information on the current version of SewerCAD. This topic, which takes the place of a ReadMe file, may include information about new features, tips, performance tuning, and other general information.
- **Services / Contents** - Connects you to the Haestad Methods Services page, where you can access information on our products and Continuing Education.

- **Services / Multimedia CD** - Opens a comprehensive demonstration of related Haestad Methods' software products.
- **Services / Haestad.com** - If you are connected to the Internet, this will take you to Haestad Methods' website for product updates and other services.
- **Services / CivilProjects.com** - If you are connected to the Internet, this will take you to our Civil Projects website, which was created as a special service for our friends and clients in the Civil Engineering community. This site contains RFPs and RFQs to help you find the work you are looking for.
- **Welcome Dialog** - Opens the **Welcome Dialog**, which is normally shown at program startup.
- **Tutorials** - Provides access to the interactive tutorials, which guide you through many of the program's features. Tutorials are a great way to become familiar with new features.
- **Using SewerCAD** - Opens a help topic with an introduction to SewerCAD and related elementary information.
- **How Do I** - Provides instructions for common tasks you can perform within the program.
- **About SewerCAD** - Opens a dialog displaying product and registration information.

# 2.4 SewerCAD Toolbars

## 2.4.1 Toolbar Button Summaries

The toolbars are grouped based on functionality, so element creation tools are located together on the tool palette, results tools are on the tool pane, and so forth.

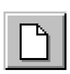


**In AutoCAD mode, some tools are provided by AutoCAD itself, so they are not included on the SewerCAD toolbars.**

## 2.4.2 Tool Pane Summary

The tool pane contains buttons for project management, data management, and presentation of results.

### *File Tools (Stand-Alone Only)*



**New** - Create a new project.

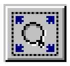
**Open** - Open an existing project.

**Save** - Save the current project.

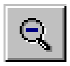
**Print Preview** - The print preview of the current view.

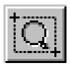
### *Zoom and Pan Tools (Stand-Alone Only)*

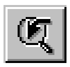
**Pan -** Pan or move around in the drawing


**Zoom Extents** - Zoom to the full extents of the drawing.


**Zoom In**


**Zoom Out**

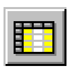

**Zoom Window** - Zoom to a selected area.


**Zoom Previous** - Zoom to the previous view.

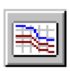
### *Calculation Tools*

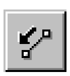
**GO** - Open the **Calculation** dialog for the current scenario.

### *Data Management and Reporting Tools*


**Tabular Reports** - Open the **Table Manager**.


**Annotation** - Annotate elements with input or output data.


**Profile** - Open the **Profile** dialog to create new profiles or view previously created ones.


**Diversion Network** - Open a plan view of the diversion network.

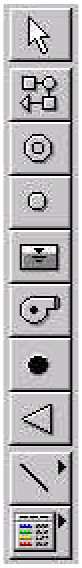

**Quick View** - Open the **Quick View** window for easy data viewing.

### *Updates and Help Tools*


**Globe** - If you are connected to the Internet, this will take you to Haestad Methods' website for product updates and other services.


**Help** - Access the on-line help system.

### 2.4.3 SewerCAD Tool Palette



The tool palette contains a **Select** tool, **Network Element** tools, and **Annotation** tools.

- The **Select** tool allows you to select elements for group editing, detailed reporting, deleting, or moving elements.
- The **Network Element** tools allow you to add elements to your network. These tools can also be used to split pipes and morph nodes.
- The **Annotation** tools can be used to add polylines, borders, and text to your drawing. You can also add a link or node color-coding legend using the **Legend** tool.

**Click a tool to select it as the active tool. In Stand-Alone mode, when a tool is selected it will be highlighted, and the cursor appearance will change to reflect the active tool.**

**In Stand-Alone mode, right-click the tool palette to access the Prototype Manager for setting the default data for each type of network element.**

### 2.4.4 Analysis Toolbar

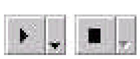
The **Analysis** toolbar displays the active scenario, provides a means for changing the active scenario, and provides access to the **Scenario Manager**. It also allows you to scroll through and animate time steps using the VCR based controls. All input and output information displayed in the tables, profiles, element dialogs, and annotation will be for the active scenario and time step shown in the **Analysis** toolbar.


You can change the current scenario from the list box, and you can access the **Scenario Manager** by clicking the folder button.

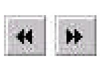
You can access the **Cost Manager** by clicking the folder with the dollar sign.

### 2.4.5 SewerCAD VCR Controls

At various locations throughout SewerCAD, such as the Analysis Toolbar, you can scroll and animate through time steps. You can do so by using the following VCR style controls:

**Play / Stop** - These buttons will stop and play an animation at the increment selected in the increment pull-down menu.



**Start Time / Stop Time -** These buttons will proceed to the first and final time step respectively.

**Decrement / Increment** - These buttons will proceed to the previous or next time step based on the increment selected in the increment pull down menu.

**Time Selector** - This pull-down menu allows you view a particular time step.

**Increment Selector -** This pull-down menu allows you to select the increment by which to scan through animations, and individual time steps.

By clicking the down arrow next to the **Play** button you can access the following **Animation Options**:

- **Animation Delay -** Opens a dialog that allows you to set the delay between animated frames.
- **Animate All Windows** - If this option is selected, then every window capable of being animated will animate when the play button is clicked. If the option is not selected then only the current window will animate.

## 2.4.6 Other Toolbar Buttons

Some of the following toolbar buttons appear on secondary windows, such as the **Print Preview** window and the **Profile** window, available throughout the program:

- File
- Print
- Print Preview
- Copy to Clipboard
- Undo
- Redo
- Options
- Page Up/Down
- Close
- Help

### Print Preview

Open a **Print Preview** on the contents of the current window.


### Page Up/Down

Navigate between pages of a multi-page report.


### File

Export the data in the current window to a file format that can be used by other applications (such as .DXF and ASCII text files).


### Copy to Clipboard

Copy data to the clipboard, where it can be pasted into most Windows-based spreadsheet, database, and word processor applications.

#### Print

Print the contents of the current window.


#### Undo

Undo the previous action.

#### Redo

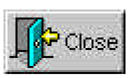
Redo the previously undone action.

#### Options

Options vary depending on the context. They may include things such as printer setup, or graph options for the current window.


#### Close

Close the current window.




## 2.5 The Status Bar

### 2.5.1 General Status Information

General status information includes messages that relate to the user's current activities. These messages consist of information such as pull-down menu command descriptions, and indications regarding the progress of an executing command.

### 2.5.2 DXF Background Status

This area of the status bar simply indicates whether a .DXF background is currently visible for the active project.

### 2.5.3 Cursor Location

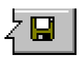
The status bar displays the current X and Y (or Northing and Easting) coordinates for the cursor's position within the drawing pane.

### 2.5.4 Calculation Results Status

In Stand-Alone mode, if the current calculation results are out-of-date or otherwise invalid, an indicator will appear in the status bar that signifies that the results no longer match the state of your input data. If the results are currently valid, no such indicator will appear.


### 2.5.5 File Status

If changes have been made since the last time the project file was saved, an image of a diskette appears in the status pane. If the file is currently in a saved state, no such image will appear.



Notes

# Chapter 3

# Quick Start Lessons

## 3.1 Overview

The purpose of Chapter 3 is to provide step-by-step lessons to get you familiar with some of the features and capabilities of SewerCAD. The lessons serve as a means to get you started exploring and using the software. We have included sample files located in your **Haestad\SWRC\Lesson** directory for you to explore and experiment with. Don’t forget to run our online tutorials, and, if you need help, press **F1** (or right click) to access our context sensitive on-line help.

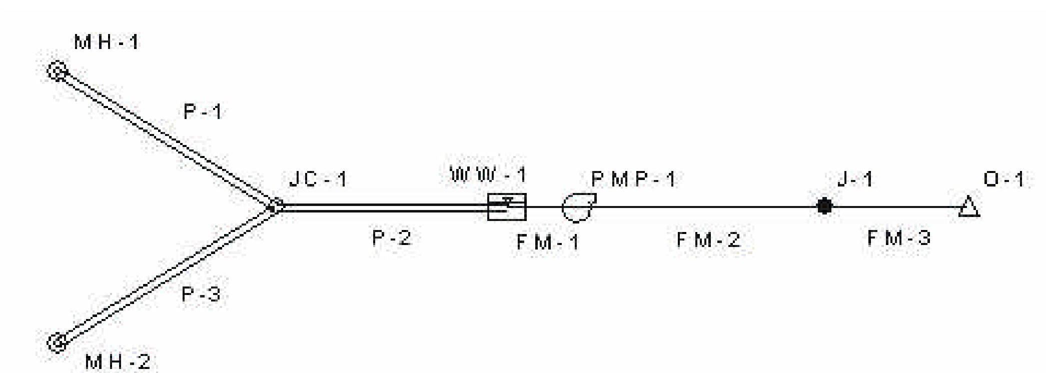
**In order to follow these tutorials, you can either do them in sequence, since each tutorial uses the results of the previous ones, or start lesson 2, 3, 4, and 5 with the files located in Haestad\SWRC\Lesson.**

## 3.2 Lesson 1 - Creating a Schematic Network

SewerCAD is an extremely efficient tool for laying out a sanitary sewer model. It is easy to prepare a schematic model and let SewerCAD take care of the link-node connectivity.

You do not need to be concerned with assigning labels to pipes and nodes, because SewerCAD will handle this internally. When creating a scaled drawing, pipe lengths are automatically calculated from the position of the pipes’ start and stop nodes on the drawing pane. Since this example is a schematic (not scaled) layout, you will need to enter the pipe lengths.

In this lesson we will layout and analyze the following schematic network.




**If, at any time during this lesson, the program asks "Do you wish to reset all calculated results to N/A?" click NO.**

### *Part 1 - Creating a New Schematic Project File*

AutoCAD specific

Stand-Alone specific

Start SewerCAD Stand-Alone. If the **Welcome to SewerCAD** dialog appears, click the **Create New Project** button. If it does not appear, choose **New** from the **File** pull-down menu. Enter a file name such as 'Lesson.swr' for your project and click **Save**.

**In Stand-Alone mode, the Welcome dialog can be obtained by selecting Tools / Options from the pull-down menus and choosing the Global tab. Set the Welcome Dialog field to Show Welcome Dialog on startup.**

**The lessons are based on a model with 6 pipes. This was done in order to illustrate the different element types and different concepts in the network layout within SewerCAD. If you have a smaller version of the product you can delete one of the manholes before calculating. To do so simply select the manhole and press the "Delete" key. The concepts will be the same.**

Start SewerCAD for AutoCAD and choose **New** on the **File** pull-down menu to create a new file. Click **No** when prompted to save the current drawing. In the **Create New Drawing** dialog, click the **Use a Template** button. In AutoCAD r14 select the SWRC14.dwt. In AutoCAD 2000 select the SWRC15.dwt file.

If you are in AutoCAD 2000 and you get prompted at the command line for a template instead of getting the **Create New Drawing** dialog, click on the **Escape** button. Type *options* at the command prompt and on the **System** tab, set the **Show Startup** dialog toggle to **ON**. When asked if you would like to setup the project, click **Yes**. If you are in AutoCAD 2000i and the **AutoCAD Today** window comes up, select the **Create Drawings Tab** and select the SWRC15.dwt template.

The remaining commands are the same in Stand-Alone and AutoCAD mode.

- Enter information about the project (optional). Click the **Next** button.
- Choose your desired settings. For this lesson, use the program default values. Click on the **Next** button.
- Select the **Schematic** radio button located under the **Drawing Scale** option. Click the **Next** button to accept the rest of the default values.
- The element prototype buttons allow you to set default values for each element type. We will use the default prototype values in this lesson. Click the **Finished** button.

### *Part 2 - Laying Out the Network*

In this example we will use Metric units. Before laying out any element, select **Tools / Options** from the pull-down menu and choose the **Global** tab. Set the global unit system to **System International**, and click **OK**.

To draw the skeletonized sewer network shown previously, select the **Pipe Layout** tool from the toolbar. Then move the cursor onto the drawing space and click once to place a manhole to represent the manhole labeled MH-1. Right-click and select wet well from the pop-up menu. Move the cursor to the approximate location of the wet well, WW-1, and click once to place it. Now, place the pump, pressure junction, and the outlet by right clicking, selecting the appropriate element from the pop-up menu, and then clicking once to place each element.

Wet wells represent the transition point between the gravity system and the pressure system. Similarly, manholes and junction chambers can represent the transition from a pressure system to the gravity system. SewerCAD automatically creates either gravity pipes (depicted by parallel lines) or pressure pipes (depicted by a single line) depending on the pipe's upstream and downstream nodes.

Place manhole, MH-2, using the **Pipe Layout** tool. Right-click and select **Junction Chamber** from the pop-up menu. SewerCAD allows you to split any pipe in two. To insert the junction chamber, click the middle of pipe P-1. A dialog will pop up asking whether you wish to split the pipe and insert a junction chamber. Click **Yes**. Right-click and select **Done** from the pop-up menu to terminate the pipe layout command. Click JC-1 and drag it down so your network matches the layout shown above.

### *Part 3 - Entering Data*

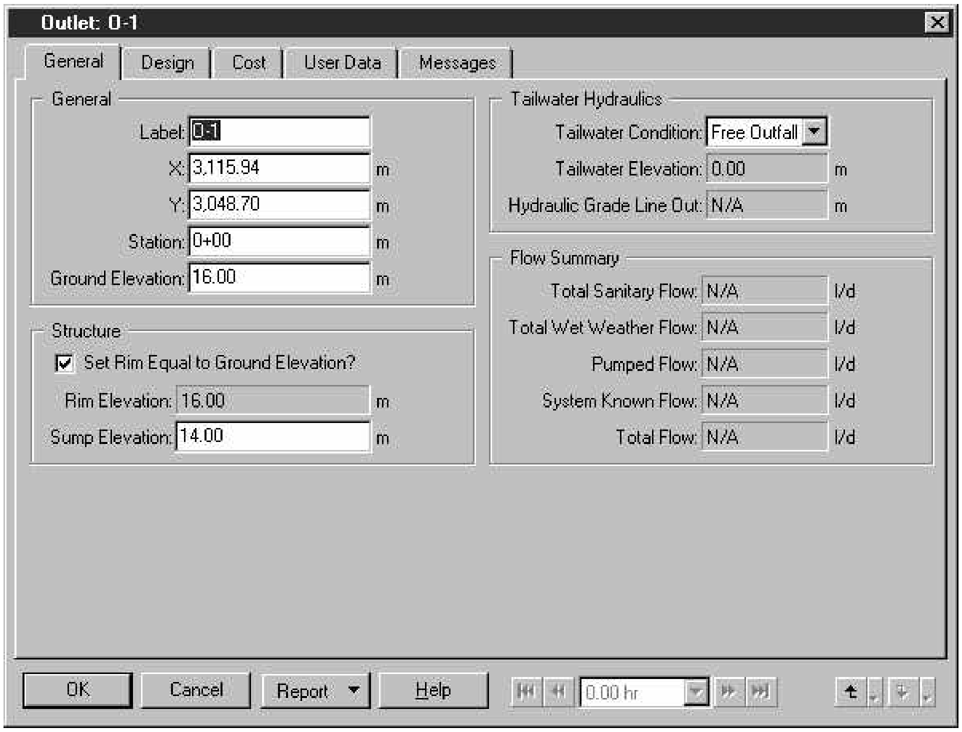
There are five ways to enter and modify element data in SewerCAD:

- **Dialogs** - You may use the **Selection** tool and double-click an element (single click in AutoCAD) to bring up its editor. In AutoCAD, click the element once with the **Selection** tool to open the element's editor.
- **Quick View Window** - Click the **Quick View Window** button in the main toolbar. You can then select an element and modify any of the white fields under the **Input** tab.
- **FlexTables** - You may click on the **Tabular Reports** button to activate dynamic tables that allow you to edit and display the model data in a tabular format. You can edit data as you would in a spreadsheet.
- **Database Connections** - The database connection feature allows you to import and export element data directly from external sources such as Excel spreadsheets, GIS, Jet Databases like Microsoft Access, and many others. This is further explained in the chapter on database connections.
- **Alternative Editors** - Alternatives are used for entering data for different "W*hat If?"* situations for use in Scenario Management. This is covered extensively in the Scenario Management chapter and Lesson 3.

### *Part 4 - Entering Data through Dialogs*

To access an element's dialog in Stand-Alone mode and AutoCAD 2000i, simply double-click the element with the cursor. In AutoCAD 2000 and R14, first click on the **Selection** tool on the toolbar, then click the element whose attributes you wish to modify.

Open the editor of the outlet, O-1, and select the **General** tab. Simply enter the data including ground elevation, rim elevation, and sump elevation as outlined in the following Outlet Data table. If the **Set Rim to Ground Elevation** box is checked, SewerCAD will automatically set the rim elevation to the ground elevation. Finally, select **Free Outfall** from the **Tailwater Conditions** choice list.




Outlet Data

| General Tab | | | | |
|---|---|---|---|---|
| Outlet | Ground Elevation (m) | Rim Elevation (m) | Sump Elevation (m) | Tailwater Condition |
| O-1 | 16 | 16 | 14 | Free Outfall |

Click **OK**. All other elements can be modified the same way.

Now enter the data for the manholes and the junction chamber as outlined in the Manhole data and Junction Chamber data tables below. Keep in mind that the headlosses are accessed by clicking the **Headlosses** tab at the top of the dialog. Select **Standard** from the list of available headloss methods in the **Headloss Method** field. Then enter the headloss coefficient for each structure.

Manhole Data

| General Tab | | | | | Headlosses Tab | |
|---|---|---|---|---|---|---|
| Manholes | Ground Elevation (m) | Rim Elevation (m) | Sump Elevation (m) | Diameter (m) | Headloss Method | Headloss Coefficient |
| MH-1 | 11.1 | 11 | 9 | 1 | Standard | 0.25 |
| MH-2 | 11.1 | 11.1 | 9 | 1 | Standard | 0.25 |

Junction Chamber Data

| General Tab | | | | | Headlosses Tab | |
|---|---|---|---|---|---|---|
| Junction Chambers | Ground Elevation (m) | Top Elevation (m) | Bottom Elevation (m) | Structure Diameter (m) | Headloss Method | Headloss Coefficient |
| JC-1 | 12 | 11 | 9.2 | 1 | Standard | 0.5 |

Open the element editor for the wet well, WW-1. Under the **General** tab, enter the station and ground elevation for the wet well given in the Wet Well Data table below. Click the **Section** tab to enter in the wet well's geometric characteristics, which are given in the Wet Well Data table below. Use the default values for the other parameters. Click **OK** to exit the dialog.

Wet Well Data

| General Tab | | | Section Tab | | | | | |
|---|---|---|---|---|---|---|---|---|
| Wet Wells | Station (m) | Ground Elevation (m) | Section | Max Elevation (m) | Initial Elevation (m) | Min Elevation (m) | Base Elevation (m) | Diameter (m) |
| WW-1 | 0+00 | 10.5 | Constant Area | 10 | 8 | 6 | 6 | 3 |

Open the element editor for the pump, PMP-1. Select **Standard (3 Point)** from the **Pump Type** choice list. Enter the pump elevation and the discharge curve as given in the Pump Data table below. Also, notice the pump has an upstream pipe and a downstream pipe to define the direction. If the pump is ever going in the wrong direction, simply click the **Reverse** button to switch it. In this example, the upstream pipe should be FM-1 and the downstream pipe should be FM-2. Click **OK** to exit the dialog.

Pump Data

| General Tab | | |
|---|---|---|
| Pump | Elevation (m) | Pump Type |
| PMP-1 | 7.8 | Standard 3 Point |

| General Tab | | |
|---|---|---|
| | Head (m) | Discharge ($m^3$/min) |
| Shutoff: | 22.67 | 0 |
| Design: | 17 | 24 |
| Max. Operating | 0 | 48 |

Open the element editor for the pressure junction, J-1. Enter the value for elevation as given in the Pressure Junction Data table below. Click **OK** to exit the dialog.

Pressure Junction Data

| General Tab | |
|---|---|
| Pressure Junction | Elevation (m) |
| J-1 | 14.2 |

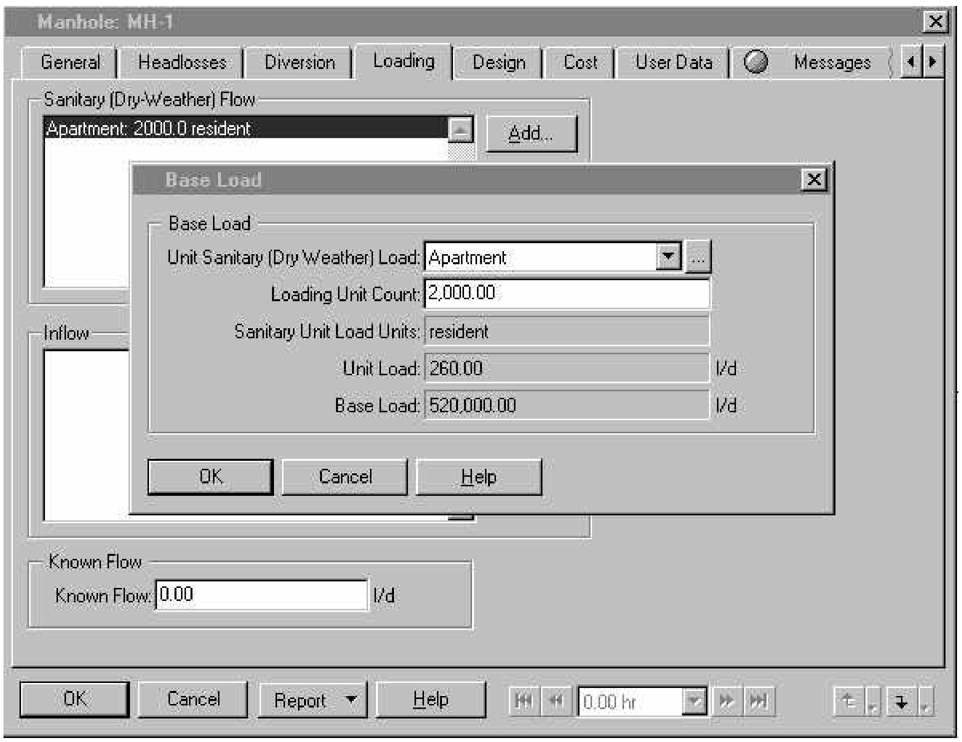
### *Part 5 - Steady State Loading*

In SewerCAD, loading is categorized as either a sanitary load or a wet weather load. Sanitary (dry weather) loads occur independent of the weather, such as wastewater from a subdivision. Wet weather loads such as pipe infiltration and inflow at nodes are directly related to the rainfall in the area. This part of the lesson deals with sanitary loads.

Loads can be applied to manholes, wet wells, and pressure junctions. The program has been designed so that all loads are designated at a node using the same procedure. To access an element's loading data, open the editor for the node of interest and click the **Loading** tab at the top of the dialog.

SewerCAD defines loads by **Unit Sanitary (Dry Weather) Load** and the **Loading Unit Count**. The Unit Load represents the amount of load per a given unit. For example, in average income housing, each resident contributes 280 l/d to the sanitary sewer. The Loading Unit Count would be the number of units. Say 40 residents live in a subdivision of average income housing. The total load would be the Unit Load multiplied by the Loading Unit Count. Thus, the total load is 40 residents * 280 (l/d)/resident which equals 11,200 l/d.

Open the editor for **MH-1** and click the **Loading** tab at the top of the dialog.




Click the **Add** button next to the **Sanitary (Dry-Weather) Flow** pane. In the **Edit** dialog, select **Unit Load - Unit Type & Count** as the **Load Definition**. Notice you can also enter in a hydrograph as a sanitary load. We shall explore time based loading further in lesson 5 on running EPS simulations. Click **OK**. In the **Base Load** dialog that appears, select **Apartment** from the **Unit Sanitary (Dry Weather) Load** pull-down menu. Then enter a **Loading Unit Count** of 2000. Click **OK** to return to the **Loading** tab on the Manhole editor.

Loading Data

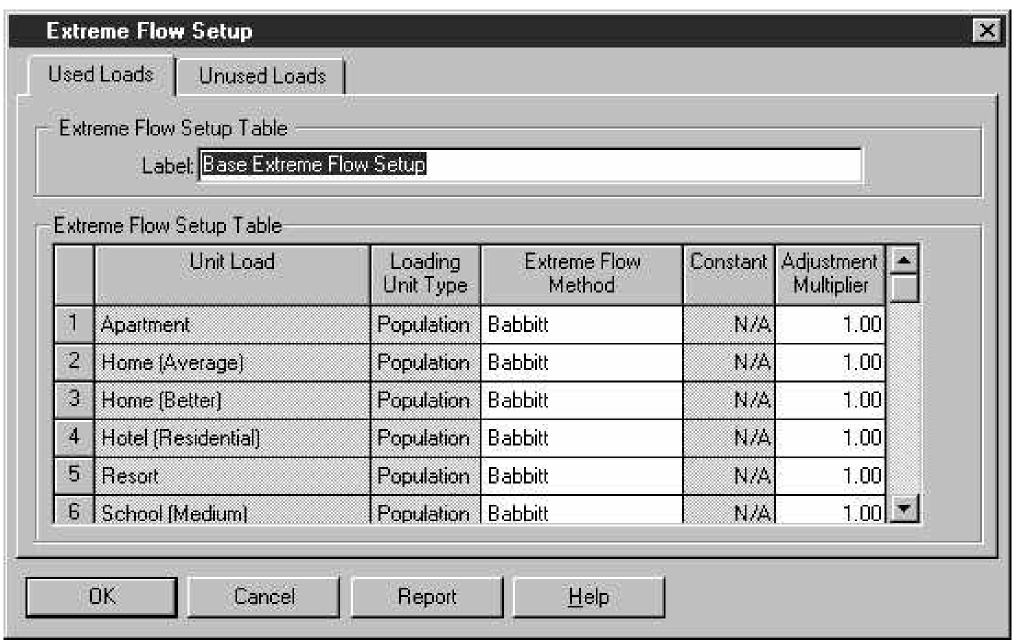
| Loading Tab | | | |
|---|---|---|---|
| Node | Unit Dry Weather Load | Loading Unit Count | Loading Unit |
| MH-1 | Apartment | 2000 | resident |
| | Home(Average) | 3000 | resident |
| | Home(Better) | 2000 | resident |
| | | | |
| MH-2 | Resort | 2000 | guest |
| | Hotel (Residential) | 1000 | guest |
| | | | |
| WW-1 | Theater | 200 | customer |
| | Shopping Center per Employee | 60 | employee |
| | | | |
| J-1 | School (Medium) | 500 | student |

Fill in the rest of the loads for MH-1 as outlined in the Loading Data table above, entering them into successive rows as they are created on the table. Click **OK** to exit the dialog.

After you have completed supplying the loading for MH-1, apply loads to MH-2, WW-1, and J-1 exactly the same way. All the loading data is outlined in the Loading Data table above.

### *Part 6 - Extreme Flow Factors*

After all loads have been applied you can specify how those average loads relate to the peak load. This is done through the **Extreme Flow Setup Manager**, which is accessed by selecting **Analysis / Extreme Flows** from the pull-down menus. In the **Extreme Flow Setup Manager** double-click the **Base Extreme Flow Setup**. In the **Extreme Flow Setup** dialog you specify which Extreme Flow method is applied and any associated constants and adjustment multipliers.

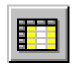



Under the **Used Loads** tab the Unit Loads currently used in the model are presented. If you wanted to apply Extreme Flow Methods to unit loads that have not been used yet, you would click the **Unused Loads** tab.

To apply an extreme flow method to a unit load, simply click the field under the **Extreme Flow Method** column and select the method you wish to use from the choice list. For this example, apply the Babbitt equation to all the used unit loads. You may speed this up by right clicking anywhere in that column, selecting **Global Edit…** from the pop-up menu, selecting **Babbitt** from the **Extreme Flow Method** drop-down box, and clicking **OK**. In this case, we wish to use the peaking factor calculated by the Babbitt method without any adjustment. Therefore, do not alter the default adjustment multiplier of 1.

Click OK to exit the Extreme Flow Setup dialog. Click OK to exit the Extreme Flow Setup Manager.

### *Part 7 - Entering Data through the FlexTables*

Often it is more convenient to enter data for similar elements into a tabular form rather than to individually click every element, enter the data into the dialog, and then click the next element. To access the tabular report, click the **Tabular Reports** button  on the toolbar.

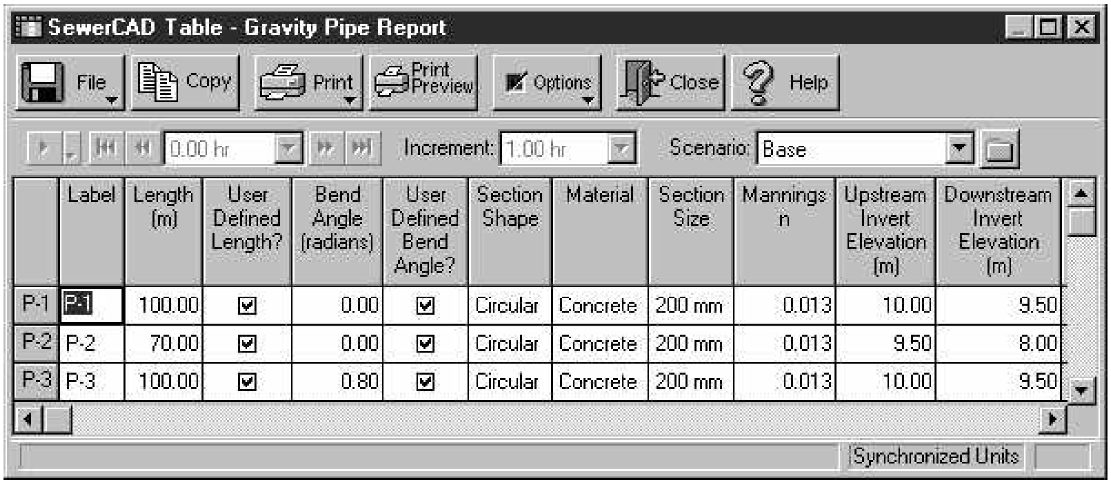
Click the **Gravity Pipe Report** and click **OK**. Enter data as you would into a spreadsheet. The yellow fields are not editable and the white fields are. For each of the three gravity pipes enter the upstream and downstream inverts, the section size, the section type, and the pipe material as outlined in the Gravity Pipe Data table below. Leave other data to default values. The gravity pipes may not be in alphanumeric order in the table. To sort the table by pipe label, right-click the **Label** column heading. Select **Sort / Ascending** from the pop-up menu that appears. You may want to maximize the window to be able to view it better.

The pipe lengths and bend angles are either calculated from the pipe orientation and position on the drawing space or they are user-defined. To make the values user-defined; check the boxes in the **User-Defined Length** and **User-Defined Bend Angle** columns of the FlexTable. This makes the bend angle and length fields editable. Now enter the length and bend angles for the gravity pipe given in the Gravity Pipe Data table below.

Gravity Pipe Data

| Gravity Pipe | Length (m) | Bend Angle (radians) | Section Shape | Material | Section Size | Upstream Invert Elevation (m) | Downstream Invert Elevation (m) |
|---|---|---|---|---|---|---|---|
| P-1 | 100 | 0 | Circular | Concrete | 200 mm | 10 | 9.5 |
| P-2 | 70 | 0 | Circular | Concrete | 200 mm | 9.5 | 9.1 |
| P-3 | 100 | 0.8 | Circular | Concrete | 200 mm | 10 | 9.5 |

The default roughness factor is based on the material chosen. If you want to add a column in order to see or change the roughness factor being used, you can easily add this field to the FlexTable. First, click the **Options** button at the top of the table and select **Table Manager** from the pull-down menu. Highlight **Gravity Pipe Report**, click the **Table Management** button, and select **Edit**. Find the **Mannings n** column in the **Available Columns** list and double-click it. **Manning's n** will show up at the end of the **Selected Columns** list. Click **OK** to exit the **Table Setup** dialog. Click **OK** on the **Table Manager** dialog to return to the **Gravity Pipe** table. The Mannings n values are displayed in the very last column of the report. Leave the other parameters set to the current values.




To go to the **Pressure Pipe** table to enter the data for pressure pipes, first click the **Options** button at the top of the **Gravity Pipe** table and select **Table Manager** from the pull-down menu. Select **Pressure Pipe Report** from the list of available tables and click **OK**. Now fill in the data as outlined in the Pressure Pipe Data table below.

Pressure Pipe Data

| General Tab | | | | | |
|---|---|---|---|---|---|
| Pressure Pipe | Material | Diameter (mm) | Upstream Invert (m) | Downstream Invert (m) | Length (m) |
| FM-1 | Ductile Iron | 200 | 6 | 7.8 | 1 |
| FM-2 | Ductile Iron | 200 | 7.8 | 13 | 200 |
| FM-3 | Ductile Iron | 200 | 13 | 14 | 100 |

There are two things to keep in mind when entering information about pressure pipes. First, invert elevations are calculated based on the elevations of upstream and downstream nodes, so they are already pre-entered. Invert elevations are only editable if the upstream or downstream node is a wet well or a gravity node. Secondly, all pressure pipes in SewerCAD are circular, so only a diameter is entered. Once all the data is entered click the **Close** button at the top of the dialog.

As you can see, all element input data can be entered through the FlexTables.

### *Part 8 - Entering Infiltration Data Into Gravity Pipes*

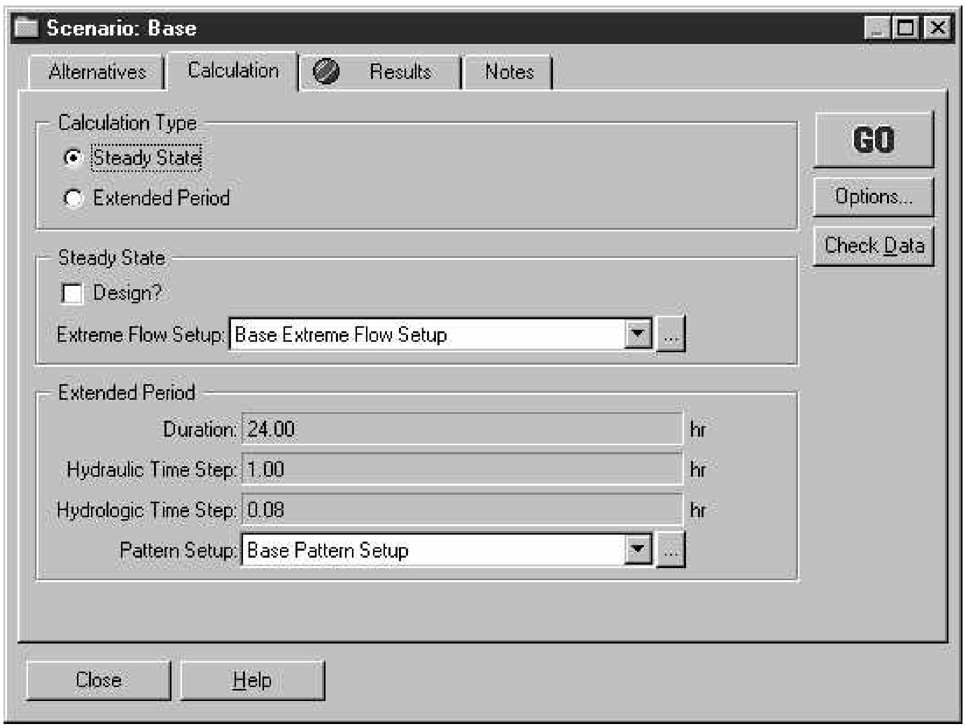
To account for infiltration into gravity pipe P-1, open the pipe's editor and click the **Infiltration** tab. Select **Pipe Length** from the choice list labeled **Infiltration Load Type**. The **Pipe Length** section will appear. In this section, select an **Infiltration Loading Unit** of "m." Then type an **Infiltration Rate per Loading Unit** of 0.25 l/d. Click **OK** to exit. Now, enter the infiltration data for the other two pipes as outlined in the Infiltration Data table below.

Infiltration Data

| Infiltration Tab | | | |
|---|---|---|---|
| Gravity Pipe | Infiltration Type | Infiltration Unit | Infiltration Rate per Loading Unit(l/d) |
| P-1 | Pipe Length | m | 0.25 |
| P-2 | Pipe Length | m | 0.05 |
| P-3 | Pipe Length | m | 0.03 |

### *Part 9 - Analyzing the System*

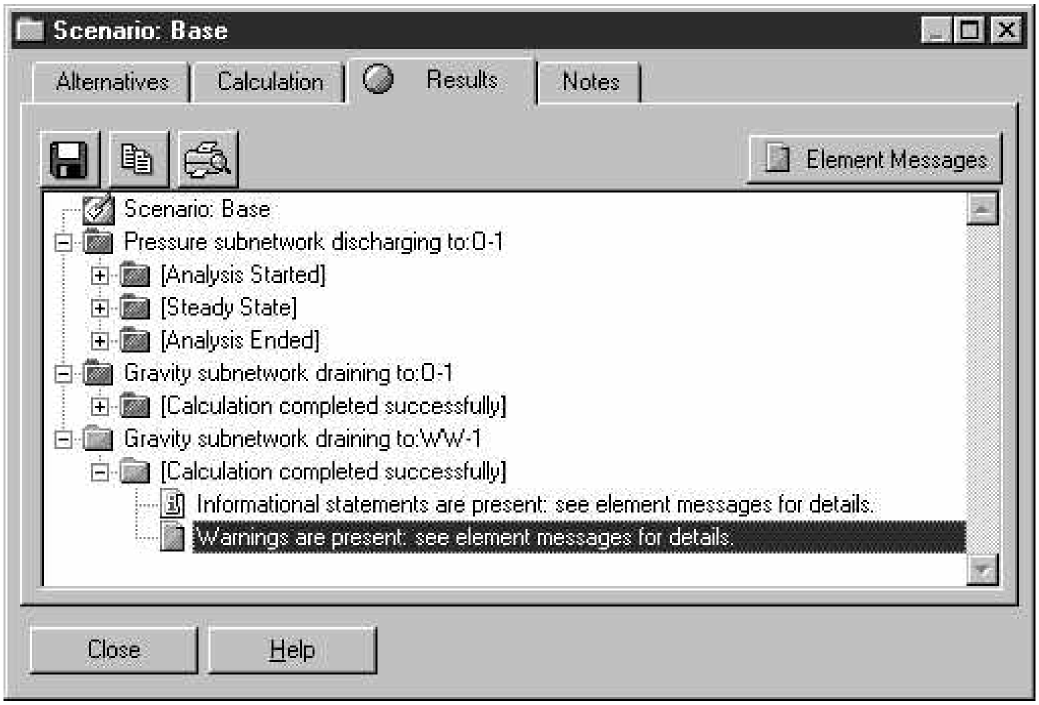
Click the **GO** button to bring up the **Calculation** dialog. Make sure that the **Calculation Type** is marked as **Steady State**.




Click the **GO** button on the dialog to analyze the model. When calculations are completed, a **Results** report is displayed.

The **Results** tab displays a summary of model results. Scroll through the summary to get an idea of the results that are given.

Notice the light displayed on the **Results** tab of the dialog. You can quickly tell if there were warnings or failures with a glance at the light. A green light indicates no warnings or failures; a yellow light indicates warnings, while a red light indicates problems. The coloring scheme is used with the folders in the results hierarchy. So if a particular folder is yellow for a calculation step, there are warnings associated with that step. Simply double click one of the folders to view the results.




Click the **Element Messages** button to display all the messages generated for individual elements during the run in a more convenient tabular format. You can exit the **Element Calculation Message Browser** by clicking the **Close** button.

Click **Close** when you are done to exit back to the drawing pane. After a model run all the calculated fields in dialogs and tabular reports will display results. See **Lesson 4** for an overview of the output options available.

As you can see from looking at the results, the performance of the gravity portion of this sewer is unacceptable for implementation because of flooding and high pressures at the junction chamber. SewerCAD's automatic design capabilities, which are outlined in Lesson 2, can quickly find an optimal solution.

Before proceeding to the next lesson, save this project. For example, save it as lessons.swr in Stand-Alone mode, or lessons.dwg in AutoCAD mode.

## 3.3 Lesson 2 - Automatic Design

This lesson will illustrate how SewerCAD can automatically design all or parts of the gravity portion of a sanitary sewer system within the design constraints set by the user. After specifying parameters such as lengths, ground elevations, and boundary conditions, SewerCAD will work to find a satisfactory design.

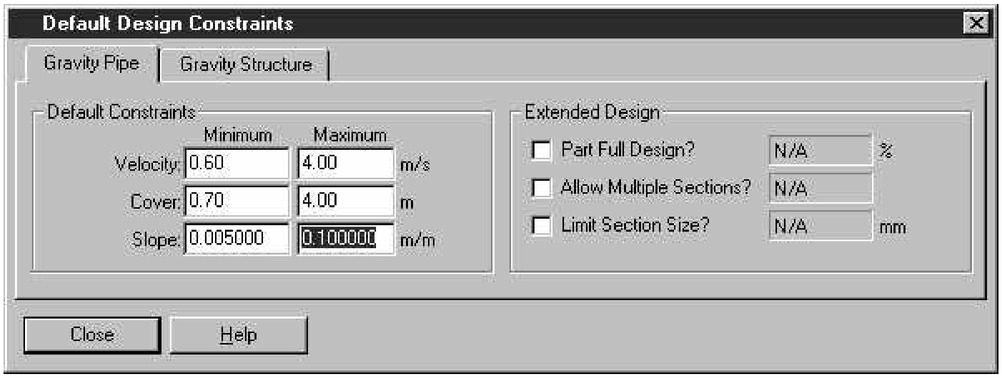
In this lesson, we will use this feature to develop a new design to replace the undersized sanitary sewer system created in Lesson 1.

**If, at any time during this lesson, the program asks, "Do you wish to reset all calculated results to N/A?" click NO.**

This lesson is based on the project created in Lesson 1. If you have not completed Lesson 1, open the project lesson2.swr (lesson2.dwg in the AutoCAD version) from the SWRC / Lesson directory.

### *Part 1: Designating Design Constraints*

SewerCAD requires parameters by which to measure the validity of a possible design. These parameters, or design constraints, can either be set locally for each individual element or they can be entered as default design constraints.




To enter the default design constraints, select **Analysis / Default Design Constraints** from the pull-down menu. On the **Gravity Pipe** tab, enter the minimum and maximum flow velocities, slopes, and covers that the newly designed pipe's characteristics should fall between. These values are listed in the Design Constraint Data table below.

Design Constraint Data

| Gravity Pipe Tab | | | |
|---|---|---|---|
| | Minimum | Maximum | |
| Velocity | 0.60 | 4.00 | m/s |
| Cover | 0.70 | 4.00 | m |
| Slope | 0.005 | 0.10 | m/m |

You could further hone the design with the Extended Design features on the right side of the dialog. Check the box next to one of three extended design criteria and enter a constraining value to have SewerCAD utilize the feature.

Click the **Gravity Structure** tab to set constraints for gravity structures. Set the **Pipe Matching** constraint to **Inverts** and the **Matchline Offset** to **0.0 m**. From this, SewerCAD knows to set the inverts of the incoming pipe at the same elevation as the invert of the outgoing pipe for the same structure. Click **Close** to exit the dialog.

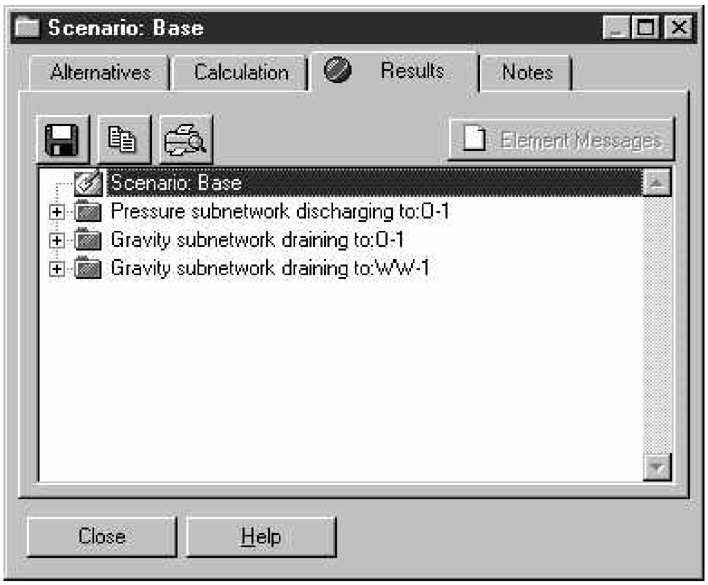
Consider that the downstream invert of Gravity Pipe, P-2, entering into the wet well, WW-1, is at a known elevation and should not be adjusted by SewerCAD's automatic design process. In this situation, you can locally specify that SewerCAD not design the downstream invert of P-2. Enter the **Gravity Pipe Editor** for P-2 by either double-clicking the pipe (Stand-Alone mode) or by first choosing the **Selection** tool and clicking the pipe once (AutoCAD mode). Click the **Design** tab. In the upper left corner of the dialog, uncheck the box next to **Design Downstream Invert**.

If the **Specify Local Constraints** box is checked, you could then fill in design constraints specific to the element whose dialog you are currently in. This feature is located under the **Design** tab for every gravity element in SewerCAD. Click **OK** to exit the **Gravity Pipe Editor**. Click **OK** to exit the dialog.

### *Part 2: Design*

To run the Automatic Design analysis, click the **GO** button on the toolbar. Click the **Design** check box in the **Steady State** section of the dialog. Then click the **GO** button to run the design. You will be asked whether you would like to create a new Physical Alternative. Alternatives are groups of data that describe a specific part of your model such as physical data, loading data, and infiltration data. Alternatives will be discussed further in Lesson 3. By clicking **Yes**, the model remembers the initial design as well as the new design for the sake of comparison. In this case click **No** and allow the model to overwrite the current physical alternative.

After the model runs, the green light on the **Results** tab indicates that there are no warnings for the design formulated by SewerCAD. SewerCAD was able to find a sewer configuration that did not cause any warning conditions and that did not violate any design constraints. Sometimes, however, it is impossible to find a solution that does not violate one of the design constraints, in which case there will be a warning to that effect presented under the **Results** tab. For more detailed information on how the program uses design constraints see Appendix B of the help. Click the **Close** button.




Save this project before proceeding to Lesson 3.

This lesson introduced one possible application of the automatic design feature. This is a powerful tool that will save you time and effort. Spend some time to learn more about this feature by experimenting with the software, and if you have any questions press the **F1** key to access our context sensitive on-line help. See Lesson 3 for more information on SewerCAD's scenario management feature.

## 3.4 Lesson 3 - Scenario Management

One of SewerCAD's many powerful and versatile project tools is Scenario Management. Scenarios allow you to calculate multiple *"What If?"* situations in a single project file. You may wish to try several designs and compare the results, or analyze an existing system using several different loading possibilities and compare the resulting profiles. A scenario consists of a group of alternatives, which are groups of actual model data. Both scenarios and alternatives are based on a parent/child relationship where child scenarios and alternatives inherit data from the parent scenarios and alternatives.

In this lesson we will use Scenario Management to set up the scenarios needed to test four *"What If?"* situations for the purpose of analyzing a new sanitary sewer system design. At the end of the lesson, we will compare all of the results using the **Scenario Comparison** tool.

**If, at any time during this lesson the program asks, "Do you wish to reset all calculated results to N/A?" click NO.**

### *Part 1 - Opening the Project File*

For this lesson we will use the system designed in Lesson 2. Click the **Open Existing File** button in the **Welcome** dialog, or select **File / Open** from the pull-down menus to bring up the **Open Project File** dialog. Open the project you saved from Lesson 2, or find lesson3.swr (lesson3.dwg in the AutoCAD version) in the SWRC / Lesson directory.

In Lesson 2 we designed the gravity portion of this system using the automatic design tool. In this lesson, we will use scenario management to model different force main designs.

### *Part 2 - Creating Alternatives*

First, we need to set up the required data sets (alternatives). An alternative is a group of data describing a specific part of the model. There are eleven alternatives: Physical Properties, Sanitary (Dry Weather) Loading, Infiltration and Inflow Loading, Known Flow Loading, Structure Headlosses, Boundary Conditions, Design Constraints, Initial Settings, Operational, Cost, and User Data. In this example, we need to set up a different physical alternative for each design trial we want to evaluate. Each physical alternative will contain different pressure pipe data.

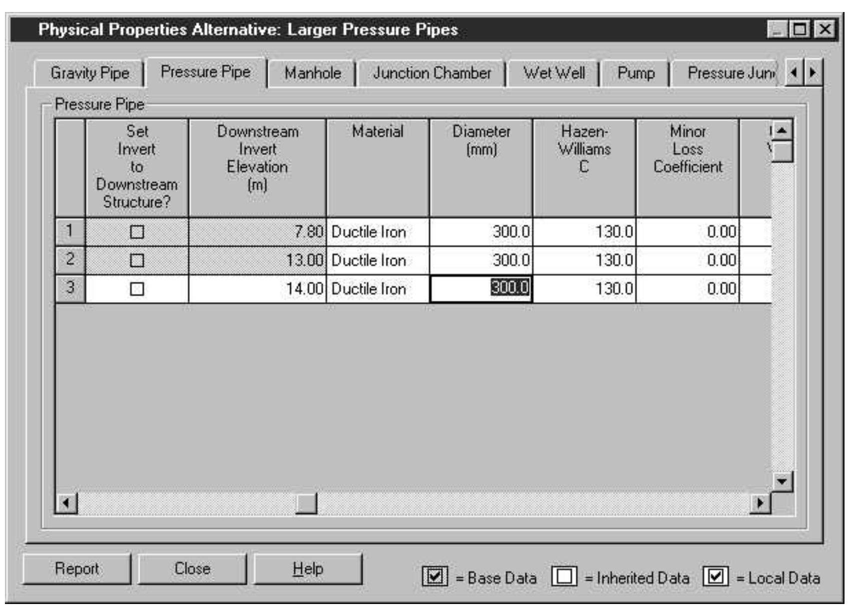
Select **Analysis / Scenarios** from the pull-down menu to load the **Scenario Manager**. Click the **Alternatives** button on the left side of the **Scenario Manager**, and select the **Physical Properties** tab.

In SewerCAD, we create families of alternatives. There are parent alternatives (base alternatives) and there are child alternatives. A child alternative inherits data from its parent. You can, however, override data inherited from the parent, making it local to the child.

Currently, there is only one Physical Alternative listed. The Base-Physical alternative contains the properties for the current undersized force mains. We would like to add a child of the Base-Physical alternative so we can inherit most of the data but change only the properties that we want to modify.

Click the **Add Child** button and enter a descriptive name such as "Larger Pressure Pipes" for the new alternative and click **OK**.

The **Physical Properties Alternative Editor** for the new alternative will appear, and contains the data that was inherited from the parent alternative. Select the **Pressure Pipes** tab at the top of the dialog. Notice the legend at the bottom describing the check boxes. It indicates, all of our data is inherited. If you change any piece of data, the check box will automatically become checked because that record is now local to this alternative and not inherited from the parent.



Set up this design trial by making the changes shown in the Pipe Alternative Data table below. Click **Close** to exit the **Physical Properties Alternative Editor** and return to the **Alternatives Manager**.

Pipe Alternative Data

| | Change From: | To: |
|---|---|---|
| | Diameter (mm) | Diameter (mm) |
| FM-1 | 200 | 300 |
| FM-2 | 200 | 300 |
| FM-3 | 200 | 300 |

Next, we will add another physical alternative for another design trial. Highlight the Base-Physical alternative and click the **Add Child** button. Enter a descriptive name for the new alternative, such as "Smaller Pump." Click **OK** to enter the **Physical Properties Alternative Editor**.

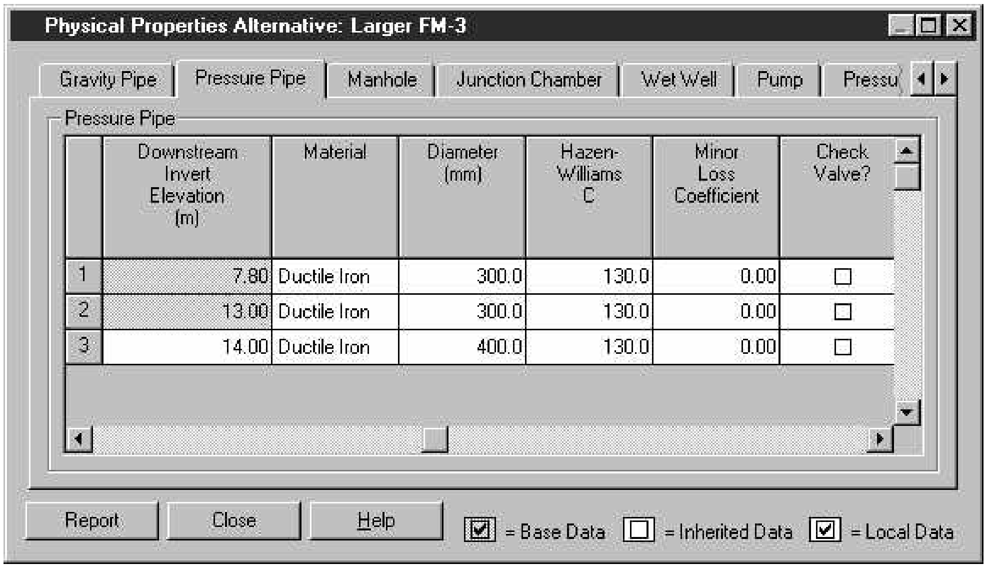
Select the **Pump** tab. For this trial, we will leave the existing system the same but with a different size pump. To change the pump curve, click the cell in the **Pump Type** column, and set it to Standard (3 point). Click the **ellipsis (…)** button to edit the pump curve. Change this design alternative by adding the data shown in the Pump Alternative Data table below and click **OK**. Click **Close** to exit the **Physical Properties Alternative Editor** and return to the **Alternatives Manager**.

Pump Alternative Data

| | Change From: | | To: | |
|---|---|---|---|---|
| | Head (m) | Discharge ($m^3$/min) | Head (m) | Discharge ($m^3$/min) |
| Shutoff | 22.67 | 0 | 20 | 0 |
| Design | 17 | 24 | 15 | 19 |
| Max Operating | 0 | 48 | 0 | 38 |

Last, we will add a Physical alternative that combines the first two design trials in the same alternative for a third design trial. Highlight the "Larger Pressure Pipes" alternative and click the **Add Child** button. Enter a name for the new alternative, such as "Larger FM-3." Click **OK** to enter the **Physical Properties Alternative Editor**.

Select the **Pressure Pipe** tab. This alternative has inherited the new pressure pipe data that we entered in the "Larger Pressure Pipes" alternative. Change the diameter of FM-3 from 300 mm to 400 mm. Click **Close** to exit the **Physical Properties Alternative** editor and return to the **Alternatives Manager**.




You now have four Physical Properties alternatives. The base alternative contains the existing system's data, while the other three contain several changes for different design trials. However, the rest of the data is the same.

Click **Close** to exit the **Alternatives Manager** and return to the **Scenario Manager**. We must now create the scenarios that will contain the Physical Properties alternatives we just created.

### *Part 3 - Editing Base Scenarios*

You are now in the **Scenario Manager**. There is always a default Base scenario that is comprised of the eleven base alternatives, currently listed in the right pane. The left pane of the **Scenario Manager** contains a list of the scenarios. Only the Base is available initially, because we have not created any new scenarios.

Alternatives are the building blocks of a scenario. A scenario is a group of the eleven alternatives and all of the calculation information needed to solve a model.

For our example, if we wish to analyze the three different design trials for the force main portion of our system, we must create a new scenario for each of the Physical Properties alternatives we created.

The first step in this process is to rename the Base scenario to a more appropriate name and set the correct calculation options. Select **Base Scenario**, click the **Scenario Management** button, and select **Rename** from the pull-down menu. The scenario name in the left pane will become editable. Type a descriptive name for the scenario, such as "Existing System" and press Enter. Next, click the **Scenario Management** button and select **Edit** from the pull-down menu. Select the **Calculation** tab. Uncheck the **Design** check box in the **Steady State** section of the dialog. Click **Close**.

### *Part 4 - Creating Child Scenarios*

The last step in setting up our scenarios is to create child scenarios. The new child scenarios will contain the Physical Properties alternatives created earlier. Highlight the base scenario entitled "Existing System" and click the **Scenario Management** button. Select **Add / Child Scenario** from the pull-down menu.

You will be prompted for a scenario name. Again, the name should be descriptive, such as "Design Trial #1." Click **OK**.

Scenarios work in families just like alternatives, except scenarios do not inherit data directly. A scenario is a group of alternatives, so a child scenario will inherit the parent's alternatives. To change the new scenario you need to change one or more of the alternatives.

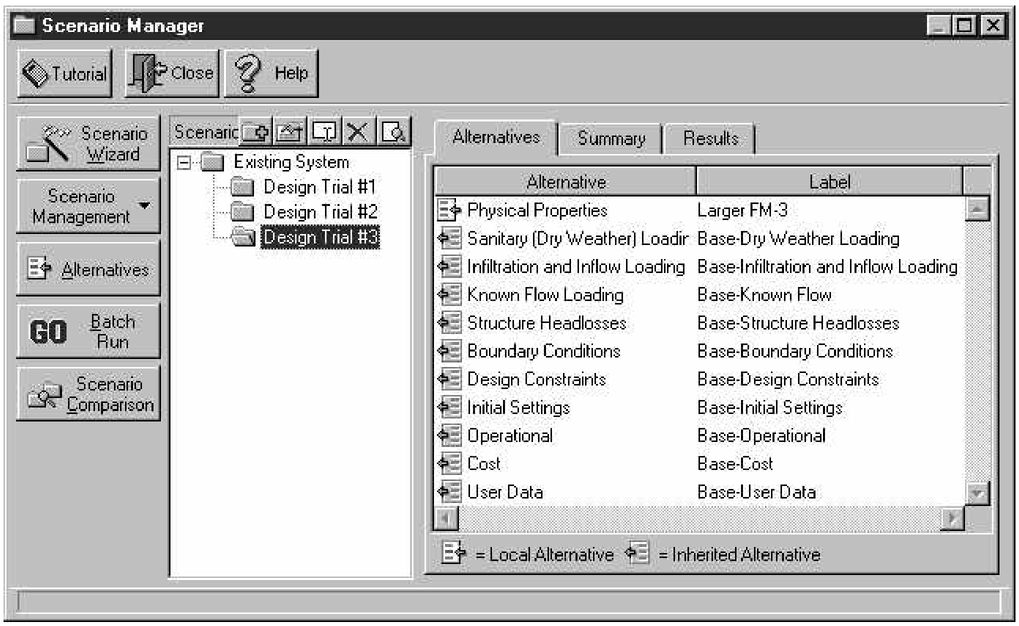
Our new child scenario initially consists of the same alternatives as its parent scenario. We want to set the Physical Properties alternative to the first alternative we created, "Larger Pressure Pipes." Click the check box next to Physical Properties to make that alternative local to this scenario. Then, from the list box, select **Larger Pressure Pipes**. Click **Close**.

Next, make sure the base scenario entitled "Existing System" is selected and click **Scenario Management / Add / Child Scenario**. Enter "Design Trial #2" into the field and click **OK**.

Again click the check box next to Physical Properties and select **Smaller Pump** from the list box. Click **Close** when you are done.

To make a third child scenario, highlight the base scenario again and click **Scenario Management / Add/Child Scenario**. Enter the scenario name "Design Trial #3" into the field and click **OK**.

Change the Physical Properties list box to **Larger FM-3**. Click **Close.**




Now we have four scenarios. The base scenario is our existing system. Each child scenario contains a different physical alternative. The first design trial resizes the pressure pipes, the second design trial resizes the pump, and the third design trial considers a different combination of pipe sizes. Now we need to calculate them.

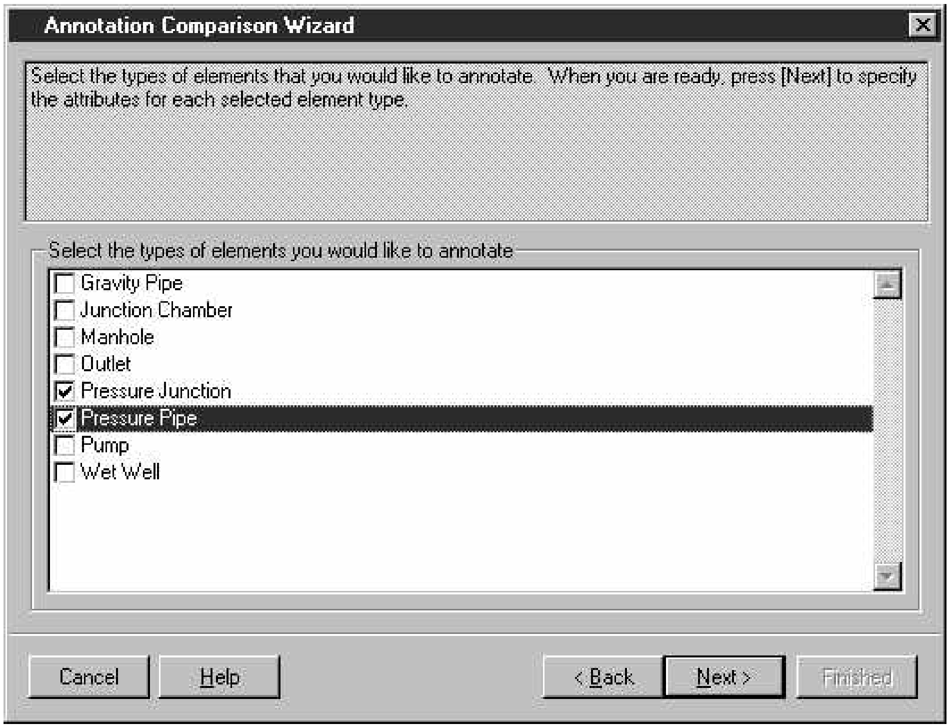
### *Part 5 - Calculate and Compare*

We are going to calculate all of the scenarios at the same time using the **Batch Run** tool. Click the **Batch Run** button on the left side of the **Scenario Manager**. Click **Select / All**, or select the check box next to each scenario, and click the **Batch** button. Click **Yes** at the prompt to run the batch for four scenarios. When it has finished computing, click **OK**.

You can see the results for each scenario by selecting it in the scenario list. Click the **Results** tab to see the selected scenario's results. We can see that each scenario is different, but what exactly is different about them? We will use the **Scenario Comparison** tool to find out.

Click the **Scenario Comparison** button to start the **Annotation Comparison Wizard**. Select the **Existing System** scenario in the first list box and the **Design Trial #1** scenario in the second list box, then click **Next**.

We will compare the results for pressure junctions and pressure pipes, so click the check box next to the Pressure Junction and Pressure Pipe and click **Next**.




Select **Pressure** from the first list box under the **Attributes** column for pressure junction annotation. You can edit the actual label in the **Mask** column. Click **Next**. Select **Velocity** from the first list box under the **Attributes** column for pressure pipe annotations and click **Next**. Verify that the annotation is correct and click **Finished**.

A plan view of the system with annotation displaying the difference between the two scenarios will appear. The difference between the two is found by subtracting Scenario 1 from Scenario 2. For example, say Scenario 1 has a total sanitary load of 4,000,000 l/d at a wet well, and Scenario 2, which represents a future scenario, has a total sanitary load of 4,500,000 l/d at the same wet well. Comparing total sanitary loads for Scenario 1 and Scenario 2 would result in annotation stating a difference of 500,000 l/d.

You can select different combinations of the four scenarios from the two list boxes and click the Update button to view the differences between the two. Or, click the **Auto Update** check box and the differences will automatically update every time you change the combination of scenarios in the list boxes. If you would like to learn more about the various results presentation methods available in SewerCAD, see Lesson 4.

Close the dialogs and save this project before proceeding to Lesson 4.

## 3.5 Lesson 4 - Presentation of Results

An important feature in all modeling software is the ability to present results clearly. This lesson outlines several of SewerCAD's reporting features, including:

- **Reports** - Displays and prints values for any or all elements in the system.
- **Element Annotation** - Dynamically presents the values of user-selected variables on the drawing.
- **Profiles** - Graphically shows how HGL and elevation vary throughout the gravity portions of the sanitary sewer.
- **Color Coding** - Assigns colors to values for a variable and applies them to the appropriate locations on the plan view for a quick diagnostic on how the system is working.

**If, at any time during this lesson, the program asks, "Do you wish to reset all calculated results to N/A?" click NO.**

### *Part 1 - Reports*

For this lesson, we will use the system designed in Lesson 3. If you did not complete Lesson 3, you may also use the file lesson4.swr, or lesson4.dwg in AutoCAD. This file is located in the SWRC / Lesson directory. After opening the file, select the "Design Trial #2" scenario from the **Scenario** toolbar. Click the **GO** button and run a regular analysis

When the **Results** dialog appears, notice that the **Results** report can be saved to a file or printed by using the buttons in the top left corner. This report displays key properties of each element on a formatted page. Click **Close**. The results for the last run can be accessed at any time by clicking the **GO** button in the toolbar and clicking the **Results** tab.

Open the **Manhole Editor** for MH-1. Click the **Report** button at the bottom of the dialog and select **Detailed Report** from the pull-down menu to view a formatted summary report of manhole MH-1's properties.

Every element has a report with the same general format, which includes the name of the calculated scenario and a series of tables describing the element's properties and results in detail. You can print this report or copy it to the clipboard using the buttons at the top of the dialog. The report will print or paste into a word processor in the exact format seen on the screen.

Click the **Close** button on the report and click **OK** to exit the **Manhole Editor**.

To print the detailed reports for several elements at one time, select **Report / Element Details** from the pull-down menus. In the AutoCAD version, the crosshair changes into a pickbox. Using the pickbox, select elements from the drawing space that you want SewerCAD to report on and right-click to bring up the list of Detailed Reports. In the Stand-Alone version, the list of elements will appear immediately. From this dialog you can select multiple elements and print them to a printer. If you wish to select multiple elements based on some criteria, click the **Select…** button to go to the **Selection Set** dialog. Click **Cancel** to return to the **Detailed Reports** dialog. You can click the **Print** button to print all of the reports for the selected elements. Click **Cancel** to exit the dialog.

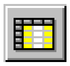
Select **Report / Element Results** from the pull-down menu. In the AutoCAD version use the pickbox to select the elements for which you want a report. Then right click once to bring up the list of Results Reports. In the Stand-Alone version, the list of Results Reports will appear immediately. From this dialog you can print or copy/paste the Results Report for any element. The Results Report contains all of the results calculated for the selected element. Again, if you wish to select a group of elements based on some criteria, click the **Select…** button. Click **Cancel** when you are done.

Select **Scenario Summary** from the **Reports** pull-down menus. This report summarizes the alternatives and options selected in the current scenario. Click **Close**.

Now select **Report / Project Inventory** from the pull-down menus. This report will tell you the total number of each type of element and the total length of pipe in the system. Click **Close**.

### *Part 2 - Tabular Reports*

Tabular Reports are extremely powerful tools in SewerCAD. These reports are not only good presentation tools; they are also very helpful in data entry and analysis. When data must be entered for a large number of elements, clicking each element and entering the data can be very tedious and time consuming. Using the tabular reports, elements can be changed using the global edit tool, or filtered to display only the desired elements. Values that are entered into the table will be automatically updated in the model. The tables can also be customized. Columns can be added or removed, or you can display duplicates of the same column with different units. The tabular reports can save you an enormous amount of time and effort.

To open a tabular report, select the **Report / Tables** from the pull-down menus or click the **Tabular Report**  button on the toolbar. Select the **Gravity Pipe Report** from the list and click **OK**.

Tabular reports are dynamic tables of input values and calculated results. White columns are input values and yellow columns are non-editable calculated values. When data is entered into a table directly, the value in the model will be automatically updated. These tables can be printed or copied into a spreadsheet program.

Two very powerful features in these tables are **Global Edits** and **Filtering**. Suppose we find that the downstream inverts of all 250 mm pipes needs to be 10 cm higher. It would be tedious to go through and re-enter every pipe invert elevation, particularly when dealing with a large system. Instead, we will use the filter tool in this example to filter out the 250 mm pipes, and the global edit tool to add 10 cm of elevation to just those pipes.

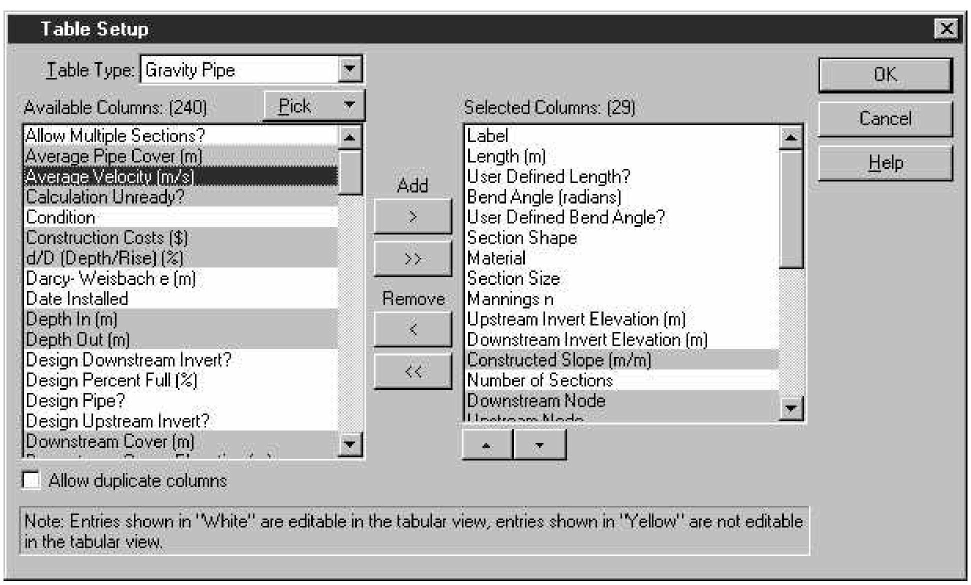
Right-click the **Section Size** column and choose **Filter / Quick Filter** from the pop-up menu. We want to filter to display only the 250 mm pipes. To do so, set the **Column** field to **Section Size**, set the **Operator** to **=**, and set the **Value** field to 250 mm. Click **OK**.

Now we will use the **Global Edit** tool to modify all of the rows in the table. Right click the **Downstream Invert Elevation** column and select **Global Edit**. Select **Add** from the **Operation** list and enter **0.1** m into the **Global Edit** text box. Click **OK**.

To deactivate the filter, right click anywhere in the dialog and select **Filter / Reset** from the pop-up menu. Click **Yes** to reset the filter.

You may also wish to edit a table to add or remove different columns. This can be done using the **Table Manager**. Click the **Options** button and select **Table Manager**. Select the **Gravity Pipe Report** from the list and click the **Table Management** button. In the pull-down menu there is also a **New** option that would allow you to create your own table. Any tables you add will be saved for use with other projects. For now, click the **Edit** option.

Use the **[ < ]** and **[ > ]** buttons to add or remove columns from your table. For this example, select **Average Velocity** from the left and click the **[ > ]** button to add it to the list of selected columns. You can adjust the order of the columns by using the arrows, or by simply dragging and dropping items.



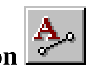

You may also wish to have two columns with the same attribute but with different units. To do this, check the **Allow Duplicate Columns** box. Then you can create the same column twice and just change the units. Click **OK** when you are done, then click **OK** in the **Table Manager**. The new table will have the added columns.

If you have made multiple columns and wish to apply separate units to the two columns, click the **Options** button and select **Use Local Units**. This option allows the tabular report to have units independent of the project and local to the table. Without this option switched on, changing the units of pressure from kPa to psi in the table would change the unit for pressure throughout the project. The **Use Local Units** option is ideal for displaying the same variable with multiple units within the same tabular report. Click **Yes** in the warning box that appears indicating that you wish to switch to local units. Then change the units on any of the numeric columns by right-clicking the column and selecting the **Properties** option. Change the units in the list box and click **OK**.

Click **Close**.

### *Part 3 - Element Annotation*

Click the **GO** button and rerun the analysis to update the results to reflect the changes in invert elevations.

Select **Tools / Element Annotation** from the pull-down menu, or click the **Annotation**  button on the toolbar.

This will activate the **Annotation Wizard**. Select the elements you wish to annotate. In this example, we will add annotation to the manholes and pressure pipes. Select these elements from the list. Click **Next**.

This dialog allows you to choose the attributes you wish to annotate for the specified element type. The **Attributes** column is used for selecting the attribute you would like to annotate. The **Mask** is a template of how the annotation will appear on the screen. The %v and %u options are added to display and control the value and units associated with the attribute. For this example, we will add annotation for the hydraulic grade line entering and exiting the manhole. In the first row of the attribute column select **Ground Elevation** from the list. In the second row of the attribute column select the **Sump Elevation** option. Click **Next**.

Add **Pressure Flow** and **Pressure Pipe Headloss** annotations for pressure pipes in the same manner described above. Click **Next**.

This is the last dialog of the **Annotation Wizard**. Check your annotations in the summary. If there are any errors, click the **Back** button to go backwards in the wizard and make any necessary changes. Click **Finished**.

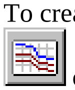
The drawing will now display all of the annotations. You can try changing the properties of an element and recalculating. The annotations will update automatically to reflect any changes in the system. You can also click and drag the annotation to move it. In the AutoCAD version, click the annotation and then click the grip to move it, or use an AutoCAD command such as Move or Stretch.

### *Part 4 - Create a Plan and Profile*

To create a plan view of the sewer system, select **Report** / **Plan View / Full View** from the pull-down menu. This will create a plan of the entire system regardless of what the screen shows, while the **Current View** option will create a plan of exactly what is displayed in the window at that moment.

The **Plan View** will be put into a separate window that can then be printed or copied to the clipboard. If you click the **Copy** button, you can then paste the plan view into a word processor. Click **Close**.

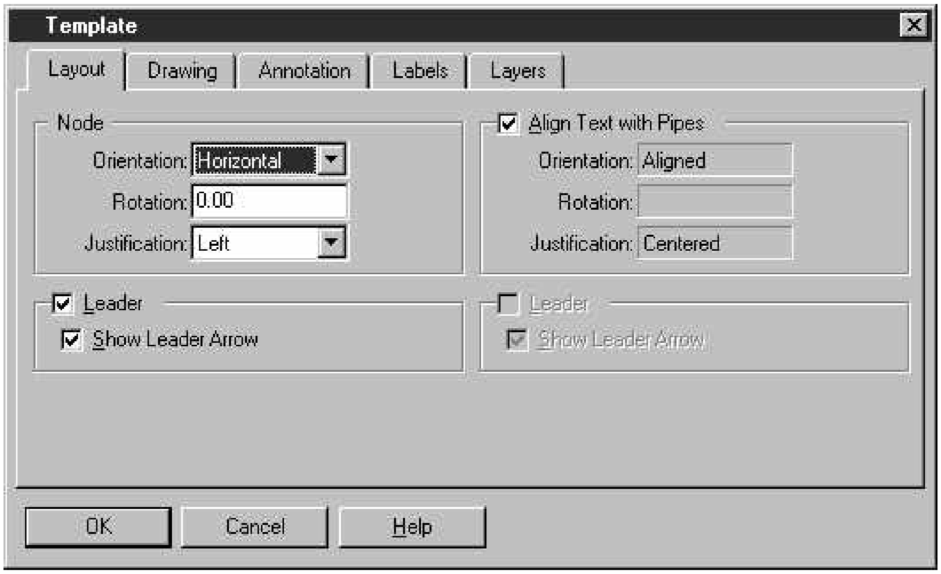
Another method that can be used to create a plan view in the Stand-Alone version that can be opened in AutoCAD is to select **File / Export / DXF File** from the pull-down menu. This will create a .DXF file of your network that you can import into AutoCAD. In AutoCAD, use plan views as a quick way to develop simple scaled views of your primary network.

To create a profile view, select **Tools / Profiling** from the pull-down menu, or click the **Profile** button  on the toolbar. This will open the **Profiles** dialog. From this dialog you can create multiple profile views for a network. For example, we can create two profiles, one from MH-2 to WW-1 and another from MH-1 to WW-1. Once the views are created you can go back to the **Profiles** to access and modify them.

Before creating a profile you can setup a profile template, which allows you to adjust many of the profile properties such as scale, label orientations, and profile annotations. By doing so, you can you reuse the same set of standards every time you create a profile, hence eliminating having to redo the same work.

Click the **Templates** button on the dialog. This will bring up the list of all available profile templates. Click the **Add** button and enter "Lessons Template" and click **OK**.

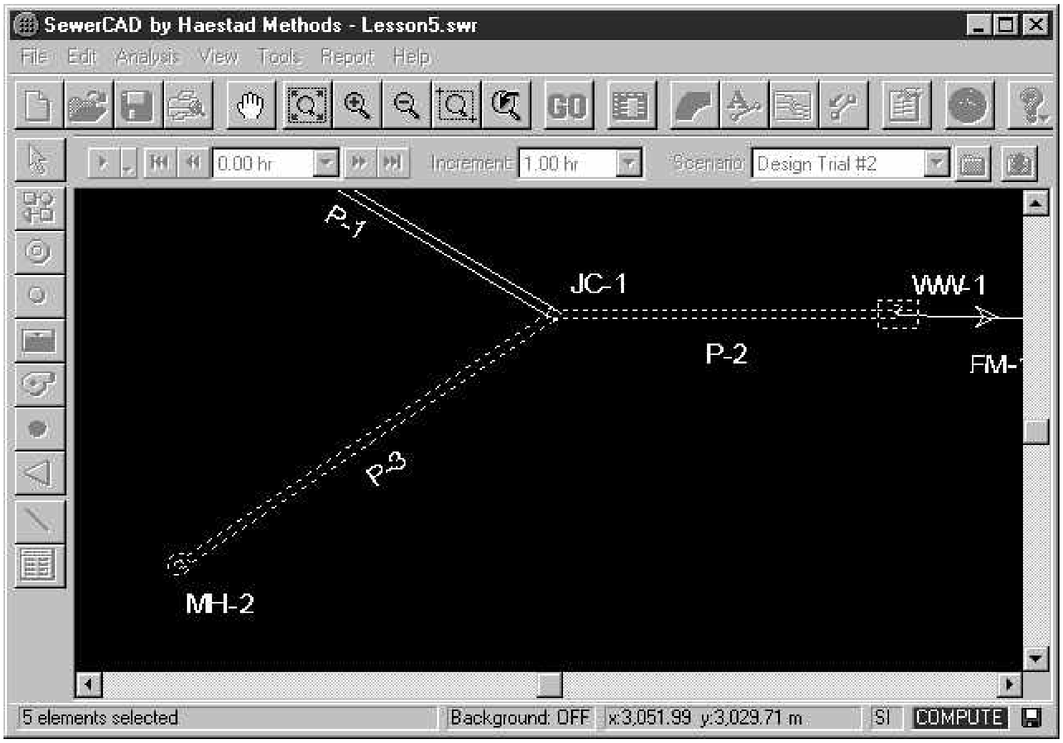
For the sake of the lesson, we are going to make two simple changes to illustrate the concept, but as you can see by scrolling through the tabs the range of customization is extensive. Click the **Layout** tab. In the **Node** section click the **Leader** check box to create a leader line from the node annotations to the nodes that they are describing.




Click the **Drawing** tab and change the **Axis Labeling** to **Both**. When this option is toggled the elevations on the Y-axis will be displayed on both sides of the profile. Feel free to play around with the other options, or adjust the annotations. Click **OK** to exit the **Template** dialog. Click **OK** a second time to return to the **Profiles** dialog.

Click the **Profile Management** button and select **Add**. In the **Label** field enter "MH-2 to WW-1," and click **OK** to open the **Profile Wizard**.

In Step-1 you select the elements to be included in the profile. Click the **Select From Drawing** button. This will open a plan view of the drawing. Note, that you can still use the zoom tools from the main toolbar. In this case click P-3 and P-2, so they are both selected. You can tell this when the lines delineating the pipes become dashed. In the following image the annotations were turned off for the sake of clarity.




When you have selected the elements right-click the mouse and select done or click the ESC key. You should see the elements you selected listed in the **Elements** section. When you are finished click the **Next** button.

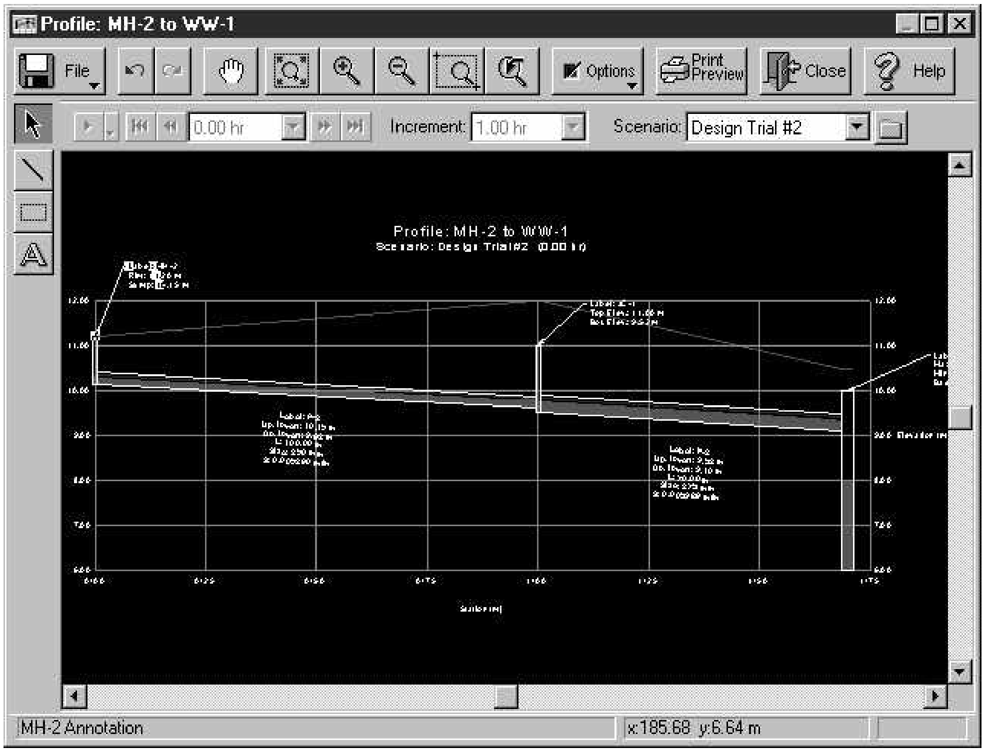
In Step-2 you can select the template to be applied to the profile. Select, **Lessons Template** from the pull-down menu and click **Next**.

This dialog allows you to set the scale for the axes, as well as the direction of the profile. These same options are accessible through the **Options / Profile Options** selection in the **Profile Window**. For this example, use the default values and click **Finished**.

The **Profile Window** will open. You can quickly move the labels to different locations by simply clicking and dragging them. Use the zoom buttons at the top to zoom in to any portion of the profile. The HGL depicts all of the flow profiles within the pipe.

You can customize annotations for the elements in the profile. To do so, click the **Options** button and select **Annotation…** Establishing the annotations is exactly the same as setting up annotations for the plan view. In this case lets add an annotation displaying the ground elevation at the wet well.

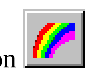
Click the **Next** button until you get to the **Wet Well Annotation** section. Select **Ground Elevation** as an attribute in the blank row, and click finished.



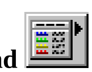

Once the profile is made, it can be exported as a .DXF file, printed, or copied to the clipboard. To export the profile as a .DXF file, click the **File** button and select **Export to DXF**. In AutoCAD, you can export profiles to the drawing by selecting **File / Export to AutoCAD**. These exported profiles do not automatically update as the model information changes.

Click **Close** when you are finished.

### *Part 5 - Color Coding*

Select **Tools / Color Coding** from the pull-down menus, or click the **Color Coding** button  on the toolbar.

The **Color Coding** dialog allows you to set the color coding for links, nodes, or both. We will color code links only in this example. From the **Color Coding** choice list, select the **Length** attribute. You can enter any range of values for length into the table. Click the **Calculate Range** button to get the minimum and maximum values for the variable displayed at the top of the dialog. Or you can click the **Initialize** button and the model will update the color coding automatically. Click **OK**.

We can add a legend to the drawing by clicking the **Insert Legend**  button and then choosing the **Link Legend** option. Click anywhere on the drawing space to place the legend. In AutoCAD, accept the defaults when prompted about scale and legend.

## 3.6 Lesson 5 - Running an Extended Period Simulation

SewerCAD has the ability to analyze time-based or extended period simulations (EPS). This lesson illustrates different aspects of setting-up and running an EPS including:

- Developing loading patterns and hydrographs
- Calculating the model

- Viewing time-based output

For this lesson, we will use the system designed in Lesson 4. If you did not complete Lesson 4, you may also use the file lesson5.swr, or lesson5.dwg in AutoCAD. This file is located in the SWRC / Lessons directory. After opening the file, select the "Design Trial #2" scenario from the **Scenario** toolbar.

**If, at any time during this lesson, the program asks, "Do you wish to reset all calculated results to N/A?" click NO.**

### *Part 1 - Entering and Applying Loading Patterns*

Loading patterns are a series of time-based multipliers that are applied to average loads, which describe how the load varies over time. In this lesson, we are going to create a loading pattern and apply it to the unit sanitary loads established with the model in an earlier lesson, and to a new Pattern Load.

To create a new loading pattern select **Analysis / Patterns** from the pull-down menus. Click the **Add** button to bring up a new **Pattern** dialog.

Enter "Lesson 5 Pattern" in the **Label** field. Then enter 0 as **Start Time**, and **Starting Multiplier** of 0.4. Then fill in the pattern table on the right-hand side of the dialog starting at hour 3.

Loading Pattern Data

| Time (hours) | Multiplier |
|---|---|
| 0 | 0.4 |
| 3 | 0.8 |
| 6 | 1.2 |
| 9 | 1.7 |
| 12 | 1.4 |
| 15 | 1.2 |
| 18 | 1.3 |
| 21 | 0.6 |
| 24 | 0.4 |

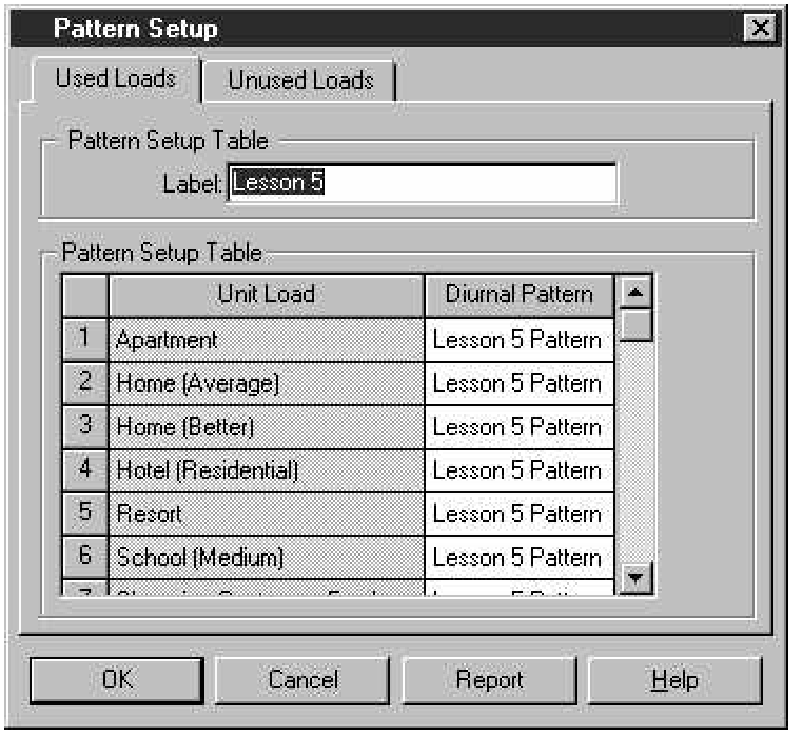
Notice that the starting and final multipliers at hour 0 and 24 are equal. The program requires this so in the case you run the simulation longer than 24 hours then the pattern can repeat itself.

Make sure that the format of the pattern is set to **Continuous**. You can see a graph of the loading pattern by clicking the **Report** button and selecting **Graph**.

To apply this pattern to the various used unit dry weather labels, select **Analysis / Pattern Setups** from the pull-down menus. The **Pattern Setup Manager** works in a similar manner to the **Extreme Flow Setup Manger**, as described in Lesson 1. In this case different patterns are applied to unit sanitary loads as opposed to extreme flow methods. As with extreme flow setups, you can create different pattern setups and associate them with different scenarios.

Click the **Add** button to create a new pattern setup. In the **Label** field enter in "Lesson 5."

In this lesson, for the sake of simplicity, we will apply the same pattern to all the unit loads. To do so, right click on the **Diurnal Pattern** column heading and select **Global Edit**. Select **Lesson 5 Pattern** from the drop-down menu and click **OK**.




Click **OK** to exit back to the **Pattern Setup Manager**. Click **OK** to exit back to the drawing.

You can also apply loading patterns to base loads set at individual hydraulic elements. To do so, enter the editor for MH-1, and click the **Loading** tab. Click the **Add** button next to the **Sanitary (Dry-Weather) Flow** pane. In the **Load Definition** pull-down menu select **Pattern Load - Base Flow & Pattern**, and click **OK**.

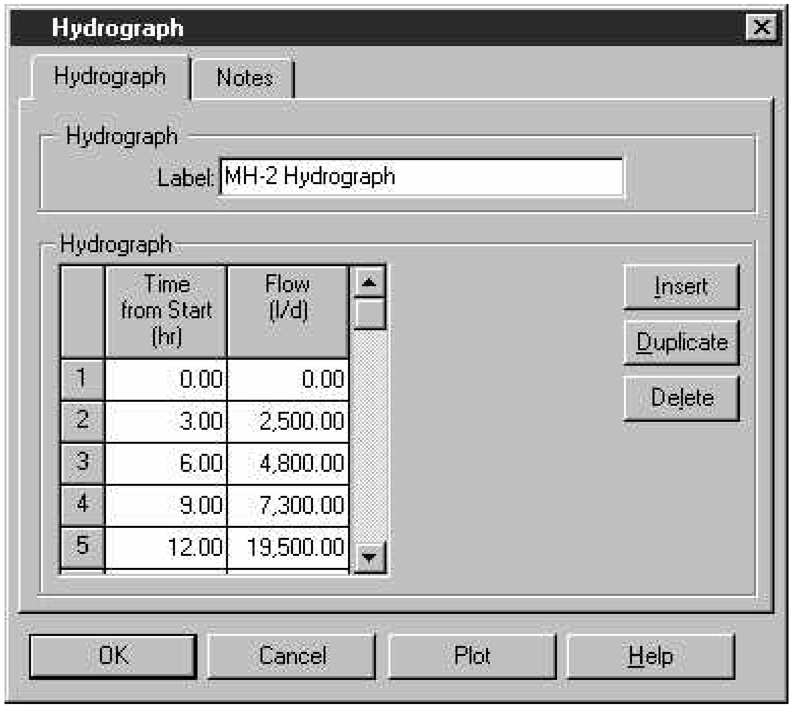
Enter in a **Base Load** of 2000 l/d, and select **Fixed** from the **Pattern** pull-down menu. Click **OK** to go back to the manhole editor, and then click **OK** in the editor dialog.

### *Part 2 - Entering Hydrographs*

SewerCAD also allows you to enter in hydrographs as a sanitary load (at manholes, wet wells, and pressure junctions), or as inflows and infiltration (at manholes, pressure junctions, wet wells and gravity pipes).

To explain the concept, we will enter in a single hydrograph as a wet-weather load at manhole MH-2. Enter the manhole editor for manhole MH-2, and click the **Loading** tab. Click the **Add** button next to the **Inflow** pane.

Select **Hydrograph - Flow vs. Time** from the **Load Definition** pull-down menu, and click **OK**. Enter "MH-2 Hydrograph" in the **Label** field. Then fill in the hydrograph with the data in the following table.




Hydrograph Data

| Time (hours) | Discharge (l/d) |
|---|---|
| 0 | 0 |
| 3 | 2500 |
| 6 | 4800 |
| 9 | 7300 |
| 12 | 19500 |
| 15 | 7300 |
| 18 | 4900 |
| 21 | 2400 |
| 24 | 0 |

Click **OK** to close the **Hydrograph** dialog, and click **OK** again to go back to the drawing pane.

### *Part 3 - Running the Analysis*

To run the EPS, click the **GO** button. In the **Scenario** dialog change the **Calculation Type** from **Steady State** to **Extended Period**. Leave the **Duration**, **Hydraulic Time Step**, **Hydrologic Time Step** set to 24, 1.00, and 0.1 respectively. Change the **Pattern Setup** from the **Base Pattern Setup** to **Lesson 5**.

Click the **GO** button in the dialog. The program will then calculate output, and when the results appear click the **Close** button.

### *Part 4 - Time Based Graphs and Tables*

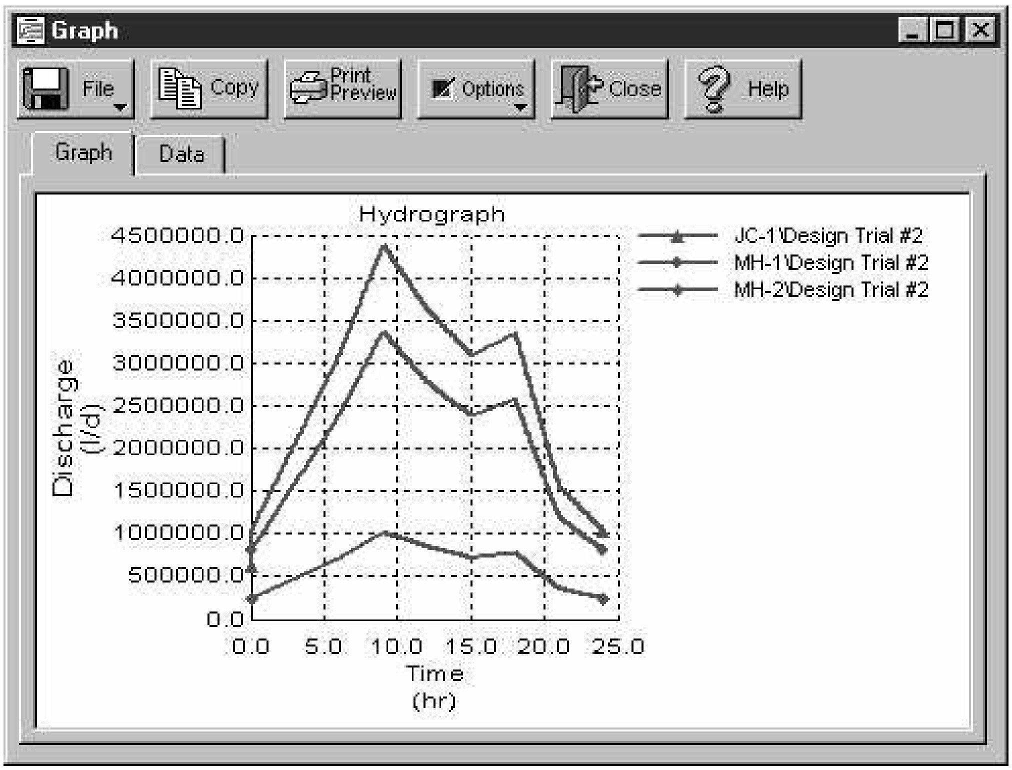
In SewerCAD you have the capability of creating time based graphs and tables for the comparison of hydrographs at multiple elements within the system.

Say we would like to compare the hydrographs generated at manholes, MH-1 and MH-2, and see the combination of the two routed hydrographs as they exits JC-1. To do so, select **Report / Hydrograph Reports**.

Under the **Scenarios** tab make sure "Design Trial #2" is checked. If multiple scenarios are checked you will see the hydrograph generated for each of the scenarios for the same element. In this case, only check the one scenario.

Click the **Elements** tab. If any elements are selected in the drawing pane they will already selected. Make sure that the three selected elements are: MH-1, MH-2 and JC-1.

When those elements are selected click the **OK** button to see the plot.




You can also see this information in a tabular format by clicking the **Data** tab. This will bring up the table. By clicking the **Copy** button the information will be copied to the clipboard you will be able to paste it into other Windows Applications such Microsoft Word or Excel.

Click **Close** to exit the plot back to the drawing pane.

You can also create other comparison graphs of attributes over time other than flow by selecting **Report / Element Graph** from the pull-down menus and then selecting the element type for which to generate a plot. As an example, select **Manhole**.

Under the **Graph Setup** tab select **Hydraulic Grade Line Out** as the **Dependent Variable**. Then select MH-1 and MH-2 under the **Elements** tab. Make sure that "Design Trial #2" is selected under the **Scenarios** tab. Click **OK** to view the graph. As you can see, the format and functionality is same as has the hydrograph plot viewed earlier. Click **Close** to return to the drawing pane.

### *Part 5 - Animations*

SewerCAD's animation tool is a dramatic, effective way of presenting and analyzing output data.

In this example, we will animate the color-coding on the main drawing pane, and the hydraulic grade line on the profile plot.

Before we animate the drawing pane, we need to color code by attribute that varies with time. In this case color code the links based on **Total Flow** attribute. Use the same procedure described in Lesson 4.

To animate the drawing pane simply click the VCR-style play button in the Analysis Toolbar. To stop the animation click the stop button which appeared upon clicking play.

If you wish to change the frame rate - click the down arrow next to the play button, and select **Animation Delay**. Increase or decrease the value depending on your preferences. Click **OK** to exit the dialog.

Profiles can also be animated in the same manner as the drawing pane. Reopen the profile created in Lesson 4, from MH-2 to the WW-1. Select **Tools / Profiling** from the pull-down menus. Select the **MH-2 to WW-1** profile and click **Open**. Simply click the play button in the profile window to start the animation and click it again to stop it.

From these five lessons, you have had a brief introduction to the capabilities of SewerCAD. Feel free to continue to play with the program. Use this model to explore and become familiar with all of the features. If you do not know what a button does, just try it.

Notes

# Chapter 4

# Starting a SewerCAD Project

## 4.1 Overview

This chapter describes how to start a new project and the files that SewerCAD creates to save your project's data. At the beginning of a project, you also need to set some global settings (accessed from the **Tools / Options** pull-down menu).

## 4.2 File Management

SewerCAD uses the .SWR extension to store all model input data, including element inputs and alternatives and scenarios, both for Stand-Alone and AutoCAD modes.

When SewerCAD runs within AutoCAD, two important files are used. The .SWR file is used to hold all model data, and a .DWG file contains all of the AutoCAD entities. This means that even a complete AutoCAD drawing corruption (or loss) will not endanger your hydraulic model data. In fact, you can even regenerate the AutoCAD modeling elements from the .SWR file!

#### *SewerCAD Backup Files*

When a .SWR file is overwritten by a save action, a backup file is created with a .SWK extension. AutoCAD also generates a backup drawing file with a .BAK extension.

#### *SewerCAD Results*

SewerCAD calculation results are stored in files with .OUT extensions for the pressure system results, and .RST extensions for the gravity system results. Since recalculating the scenarios can regenerate these results, these files do not necessarily need to be included when backing up your important files.

### 4.2.1 Multiple Sessions

SewerCAD does not support multiple sessions. SewerCAD uses a single document model, and support for multiple views has not been implemented. Therefore, do not try to open more than one session of SewerCAD at the same time, as data loss and data corruption may occur.

## 4.3 Project Management

### 4.3.1 Project Setup Wizard

The **Project Setup Wizard** can only be accessed at the start of a new project. All of the options that are edited from the Wizard, however, can be changed individually from other pull-down menus.

The **Project Setup Wizard** assists you in the creation of a new project by stepping you through many of the project-wide options, allowing you to set up most of your notes and defaults before you even create the first pipe. The areas covered by this Wizard include:

- **Project Summary** - Includes information about the project, such as the project title, the project engineer, and general comments.
- **Project Options** - Include information regarding global options, such as the desired friction method and coordinate system.
- **Drawing Options** - Include information regarding the drawing pane, such as the drawing scale, annotation multipliers, and background drawing data (for Stand-Alone mode only).
- **Prototypes** - Enable you to set default values for elements, which are used to initialize values for any new elements that are added to the project.

### 4.3.2 Project Summary

The **Summary** dialog provides a way to enter a Project Title, the name of the Project Engineer, and any significant comments (for example, the project revision history). The **Date** field defaults to the current day. To change any portion of the date, click the item to be changed (i.e: month field), then use the up and down arrows on the keyboard to set the date.

The Project Title and Project Engineer will print in the footer of reports.

To access the **Project Summary** dialog, select **File / Project Summary** from the pull-down menus.

## 4.4 Options

### 4.4.1 Global Options

The **Global Options** dialog allows you to customize the following options for this application:

- **Welcome** dialog (Stand-Alone mode only)
- Unit System
- Enter Key Behavior
- Background/Foreground Color (Stand-Alone mode only)
- Sticky Tool Palette (Stand-Alone mode only)
- Auto Prompting
- Right-Click Context Menu (This toggle exists only in AutoCAD R14; the option is automatically On in AutoCAD 2000)

To access the **Global Options** tab, select **Tools / Options** from the pull-down menus.

#### Welcome Dialog

The **Welcome** dialog appears when the program is started, and provides easy access to common tasks you may want to perform when you first start using the program. The following options are available:

- Tutorials
- Exit Program

## Unit System

Although individual units can be controlled throughout the program, you may find it useful to change your entire unit system at once, to either the **System International** (metric) unit system or the **US Customary** (English) system.

When you switch to a different unit system, you will be asked to confirm this action. If you choose **Yes**, all data will be displayed in the default units for the selected system.

> **If the file that you are editing in Stand-Alone mode is already associated with an AutoCAD drawing, be careful not to change the unit systems or the .DWG and the .SWR files may become irreversibly out of sync.**

## Enter Key Behavior

Enter Key Behavior controls which standard the **Enter** key follows during editing:

- **CUA Enter Key** - With this setting, the **Enter** key acts as it normally does for Windows applications. It is conforming to Common User Access (CUA) standards. This means that when you press the **Enter** key, it is as though you pressed the default button on the dialog. CUA Enter Key is the recommended setting.
- **Tabbing Enter Key** - With this setting, the **Enter** key behaves the same as the **Tab** key for editable fields (not buttons). This means that when you press the **Enter** key, the cursor will move to the next field in the dialog.

## Window Color

You can specify the background and foreground colors of the main graphical window in Stand-Alone mode. The foreground color is the default color that is applied to all elements symbols, pipes, labels, and annotations when no color coding is defined. These color settings also apply to the **Scenario Comparison** window, but do not apply to the **Profile** or **Graph Plot** windows.

## Sticky Tools

Sticky Tools are available in Stand-Alone mode. With **Sticky Tools** disabled, the drawing pane cursor will return to the **Select** tool after creating a node or finishing a pipe run. With **Sticky Tools** enabled, the tool does not reset to the **Select** tool, allowing you to continue dropping new elements into the drawing without reselecting the tool.

The **Sticky Tool Palette** can be turned on or off to meet your needs and preferences.

## Auto Prompting

Auto Prompting allows you to immediately enter data as elements are added to the drawing, without interrupting the layout process.

When Auto Prompting is active, the **Auto Prompting** dialog will immediately appear when you add an element to the drawing. From the **Auto Prompting** dialog, you can modify the element's default label and access the remaining input data by clicking the associated **Edit** button. Auto Prompting can also be toggled off in this dialog.

## Right-Click Context Menu Option (AutoCAD Only)

If the **Right-Click Context Menu** option is enabled, a right mouse click on a SewerCAD entity in AutoCAD R14 will activate a pop-up menu for editing or modifying the element. This functionality emulates the ability that is available in SewerCAD Stand-Alone mode. Right-clicking any other entity in the drawing will invoke standard AutoCAD right-click behavior.

**In AutoCAD 2000/2000i, this option is always available. Simply select the element in the AutoCAD drawing and right-click to obtain a pop-up menu, from which you can select Edit.**

## 4.4.2 Project Options

The **Project Options** dialog allows you to set the following essential information about your project:

- Friction Method
- Input Modes
- Pipe Length Rounding

To access the **Project Options** tab, select **Tools / Options** from the pull-down menus.

### Friction Method Theory

The **Friction Method** option enables you to select the methodology for determining flow resistance and friction losses during calculations.

Available methodologies include:

- Darcy-Weisbach
- Hazen -Williams Formula
- Kutter's Equation
- Manning's Formula

If you change the friction method after pipes have been entered into the network, the program will ask if you want to update the roughness values of those pipes. If you select **Yes**, the program will assign all pipes a new roughness that corresponds to the default roughness of the pipe material.

### Input Modes

This program supports several input modes to adjust data entry to your style or the needs of a particular project.

- **Coordinates** - Coordinates can be displayed either as X and Y coordinates, or as Northing and Easting. Whichever coordinate input mode is chosen, this method will be active everywhere within the program.
- **Hydraulic Settings** - This choice list lets you set whether values on control conditions will be input in terms of hydraulic grade or pressure. Regardless of the mode you choose for input, the program will always display values in both hydraulic grade and pressure.
- **Wet Well Levels** - This choice list lets you set whether wet well operating ranges will be input in terms of elevations (height above a datum elevation of 0) or levels (height above the wet well's base elevation).

#### *Navigation*

To access the **Input Mode** options, select **Tools / Options** from the pull-down menus, and select the **Project Options** tab.

### Pipe Length Rounding

Pipe length rounding is used to determine the level of precision desired for scaled pipe lengths. Pipe lengths will automatically be rounded according to the pipe length rounding value.

For example, consider a pipe with an actual scaled length of 35.8 meters. If the pipe length rounding value is 1.0 meters, the program will assume the pipe length to be 36.0 meters.

**This only affects the value as it appears in elemental editors, FlexTables, and so on. The actual length of the pipe figure in the drawing pane is not physically adjusted to force the pipe to a rounded length.**

**A change to the pipe rounding length is not retroactive. Therefore, it will not affect existing pipes unless the User Defined Length is toggled off and then on again in the appropriate element editor.**

### 4.4.3 Drawing Options

The **Drawing Options** dialog allows you to specify information regarding the graphical display of elements in the drawing pane, including:

- Drawing Scale
- Annotation Multipliers
- Pipe Text
- Background Drawing
- Symbol Visibility

To access the **Drawing Options** tab, select **Tools / Options** from the pull-down menus.

#### Drawing Scale

You can set the scale that you want to use as the finished drawing scale for the plan view output. Drawing scale is determined based upon engineering judgment and the destination sheet sizes to be used in the final presentation.

You may choose either schematic or scaled mode to define the horizontal and vertical distance scales.

- **Schematic** - Pipe lengths are not automatically initialized from their lengths in the drawing pane, but must be manually entered for each pipe.
- **Scaled** - Pipe lengths are determined from the lengths of the pipe elements in the drawing pane.
  - **HOR** - Horizontal scale controls the scale of the plan view.
  - **VER** - Vertical scale controls the default elevation scale (for use in profiles, for example).

Scaled or schematic mode can be set on a pipe-by-pipe basis. This is useful when scaled mode is preferred, but an exaggerated scale is needed for layout of detailed piping arrangements.

Whether the drawing is set in scaled or schematic mode automatically reflects the setting of the pipe prototype. While in schematic mode, Gravity Pipe Prototypes and Pressure Pipe Prototypes can be assigned a default length. When the drawing mode is scaled, pipe lengths do not need to be initialized from the prototype. Switching between scaled and schematic in either the **Project Options** or **Pipe Prototype** dialogs has no effect on existing pipes.

#### Annotation Multipliers

Annotation multipliers allow you to change the size of symbols, labels and annotation text relative to the drawing scale. There is not a single annotation size that is going to work well with all projects and scales, so these values should be adjusted based on your judgment and the desired look of the finished drawings.

- **Symbol Size** - The number entered in this field will either increase or decrease the size of your symbols by the factor indicated. For example, a multiplier of 2 would result in the symbol size being doubled. The program selects a default symbol height that corresponds to 4.0 ft (approximately 1.2 m) in actual-world units, regardless of scale.

- **Text Height** - The text height multiplier increases or decreases the default size of the text associated with element labeling by the factor indicated. The program automatically selects a default text height that displays at approximately 2.5 mm (0.1 in) high at the user-defined drawing scale. A scale of 1.0 mm = 0.5 m, for example, results in a text height of approximately 1.25 m. Likewise, a 1 in = 40 ft scale equates to a text height of around 4.0 ft.

- **Annotation Height** - The annotation height multiplier increases or decreases the default size of the element annotation by the multiplier indicated. The program automatically selects a default text height that displays at approximately 2.5 mm (0.1 in) high at the user defined drawing scale. A scale of 1.0 mm = 0.5 m, for example, results in a text height (to scale) of approximately 1.25 m. Likewise, a 1 in = 40 ft scale equates to a text height of around 4.0 ft.

- **Pipe Text** - Selecting the **Align Text with Pipe** box aligns the text with pipes. For more information, see the Element Annotation topic.

In AutoCAD mode if you change the **Symbol Size**, **Text Height**, or **Annotation Height** you will be prompted with the **Text Positioning** dialog, which allows you to select one of the following options when applying the scaling operation.

- **Maintain current text positions** - The current position of all the annotation will be maintained after the scale is changed.

- **Reset text to default positions** - The annotation will be repositioned to the default position calculated by the program.

On the **Text Positioning** dialog there is a check box that is labeled **Don't show this dialog again**. If you check this box you will not be prompted with this dialog until you turn it back on using one of the following commands at the AutoCAD command line.

**WTRCScaleChangeOptions** - if you are in WaterCAD

**STMCScaleChangeOptions** - if you are using StormCAD

**SWRCScaleChangeOptions** - if you are using SewerCAD

### Background Drawing (Stand-Alone mode only)

In Stand-Alone mode, a .DXF file may be used as a background image for the drawing pane.

- **DXF Background Filename** - This field enables you to specify a .DXF file to be used as the background for your project. Enter the drive, directory, and file name, or click the **Browse** button to select a file interactively.

- **Show Background** - If the background .DXF file is turned off, it will not be read from a disk or displayed in the drawing pane. If the background is not turned off, it will be read from a disk and displayed.

- **DXF Unit** - The .DXF drawing unit conversion is used when importing .DXF background files, and also when exporting a .DXF file from the project. Note that the value in this field governs the import behavior for .DXF files saved in scientific, decimal, or fractional units, but not for .DXF files saved in architectural or engineering units.

.DXF file import behavior is governed by specific factors within the .DXF file. If a file does not import as you expect, check the options used to generate it carefully. For example, try importing the .DXF back into the original program or into another program that supports the .DXF format, such as AutoCAD or MicroStation. If the file does not import into other applications, there may be an invalid or missing header, invalid elements, or other errors.

### Symbol Visibility

Symbol visibility allows you to customize the drawing by turning specific layers on or off. Each drawing layer holds a particular type of graphical element, such as labels and annotation. To remove the graphical elements of a particular layer from the drawing view, simply uncheck the appropriate boxes, which are as follows:

- **Show Labels** - The label layer holds the labels for all network elements.
- **Show Graphic Annotations** - Graphic annotation includes lines, borders and text (in Stand-Alone mode only).
- **Show Element Annotation** - Element annotation includes any dynamic annotation that is added to the project, such as through the **Annotation Wizard**.
- **Show Control Symbols** - A symbol may be displayed next to pump and valve elements with one or more controls, as defined in the **Controls** tab of the element editors.
- **Show Flow Arrows** - Arrows indicating the flow direction may be displayed after calculations have been run.

# 4.5 FlexUnits

## 4.5.1 FlexUnits Overview

FlexUnits (the ability to control units, display precision, etc) are available from almost anywhere within Haestad Methods' software, including element dialogs, FlexTables, and the FlexUnits Manager.

## 4.5.2 Field Options

Most dialogs provide access to FlexUnits to set options such as unit, rounding, and scientific notation for any field in the dialog.

To set the display options for a unitized attribute:

1. Right-click the field, and select **Properties** from the pop-up menu. The **Set Field Options** dialog will appear.
2. Set the options you want for your units.
3. Click **OK** to set the options for the field, or **Cancel** to leave without making changes.

You will be able to change the following characteristics:

- Units
- Display Precision
- Scientific Notation
- Minimum and Maximum Allowable Values

Some attributes do not have theoretical minimum or maximum values, and others may have an acceptable range governed by calculation restrictions or physical impossibilities. For these attributes, minimum and maximum allowable values may not be applicable.

**You can see the results of your changes in the preview at the top of the dialog.**

### 4.5.3 Units

Units are the method of measurement displayed for the attribute. To change units, click the choice list, then click the desired unit. The list is not limited to either SI or US customary units, so you can mix unit systems within the same project.

FlexUnits are intelligent - the units actually have meaning. When you change units, the displayed value is converted to the new unit, so the underlying magnitude of the attribute remains the same.

For example, a length of 100.0 feet is not converted to a length of 100.0 m or 100.0 in. It is correctly converted to 30.49 m or 1200.0 in.

To access set units, right-click the attribute's field and select **Properties** from the pop-up menu, or select **FlexUnits** from the **Tools** pull-down menu.

### 4.5.4 Display Precision

The precision setting can be used to control the number of digits displayed after the decimal point, or the rounding of numbers.

#### *Number of Digits Displayed After Decimal Point*

Enter 0 or a positive number to specify the number of digits after the decimal point.

For example, if the display precision is set to 3, a value of 123.456789 displays as 123.457. This works the same regardless of whether scientific notation is active.

#### *Rounding*

Enter a negative number to specify rounding to the nearest power of 10. (-1) rounds to the nearest 10, (-2) rounds to the nearest 100, and so on.

For example, if the display precision is set to (-3), a value of 1,234,567.89 displays as 1,235,000.

**Display precision is for numeric formatting only and will not affect calculation accuracy.**

To access display precision, right-click the attribute's field and select **Properties** from the pop-up menu, or select **FlexUnits** from the **Tools** pull-down menu.

### 4.5.5 Scientific Notation

Scientific notation displays the number as a real number beginning with an integer or real value, followed by the letter "**e**" and an integer (possibly preceded by a sign). Click the field to turn scientific notation on or off. A check will appear in the box to indicate that this setting is turned on.

**Scientific Notation is for numeric formatting only and will not affect calculation accuracy.**

To access scientific notation, right-click the attribute's field and select **Properties** from the pop-up menu, or select **FlexUnits** from the **Tools** pull-down menu.

### 4.5.6 Minimum and Maximum Allowed Value

Minimum and maximum values are used to control the allowable range for an attribute, and are used for validation of user input. For example, some coefficient values might typically range between 0.09 and 0.20. A frequent user input error is to misplace the decimal point when entering a value. If you enter a number that is less than the minimum allowed value, a warning message will be displayed. This helps reduce the number of input errors.

You may change this number in cases where you find the default limits too restrictive.

**These allowable minimums and maximums are only available for certain parameters.**

To access unit minimum and maximum, right-click the attributes field and select Properties from the pop-up menu, or select **FlexUnits** from the **Tools** pull-down menu.

### 4.5.7 FlexUnits Manager

The **FlexUnits Manager** allows you to set the parameters for all the units used. The dialog consists of the following columns:

- **Attribute Type** - Parameter measured by the unit.
- **Unit** - Type of measurement displayed. To change the unit of an attribute type, click the choice list and click the unit you want. This option also allows you to use both US Customary and SI units in the same worksheet.
- **System** - Set the system of units. Click the system column for the desired unit, and a button will appear. Click the button, and set the unit system to US or SI.
- **Display Precision** - Rounding of numbers and number of digits displayed after the decimal point. Enter a negative number for rounding to the nearest power of 10: (-1) rounds to 10, (-2) rounds to 100, (-3) rounds to 1000, and so on. Enter a number from 0 to 8 to indicate the number of digits after the decimal point. This feature works the same whether scientific notation is on or off.
- **Scientific Notation** - Display numbers in scientific notation. Click the field to turn scientific notation on or off. If it is turned on, a checkmark appears in the box.

**The display units can also be changed from several other areas in the program, and any changes are project-wide. For example, if length is changed from units of feet to meters, all dialogs will display length in meters. If you change the units in the dialog from meters to yards, the FlexUnits Manager will indicate that length is in yards.**

To access the FlexUnits Manager, select **Tools / FlexUnits** from the pull-down menus.

## 4.6 Quick Attribute Selector

Whenever attributes are selected such as when setting up annotations or database connections, you can select them from organized categories using the **Quick Attribute Selector** tool.

Simply click the ▸ in the attribute field to bring up the pull-down menu.

From this menu you can either select attributes from the list of available categories, or you can select from a list of the most frequently used attributes.

If you select **Frequently Used / Edit** from the pull-down menu, the **Select Field Links** dialog will appear. You can then choose all the attributes you would like to appear in the **Frequently Used** list.

Notes

# Chapter 5

# Layout and Editing Tools

## 5.1 Graphical Editor Overview

This chapter describes the various tools that are available to simplify the process of graphically or manually entering network data. These tools allow you to select elements to perform various graphical or editing operations, locate particular elements, review the network for potential problems, label or relabel elements, review your data, or define any new type of data.

## 5.2 Graphical Editor

### 5.2.1 Using the Graphical Editor

One of the most powerful features of the graphical editor, both in Stand-Alone and AutoCAD modes, is the ability to create, move, edit, and delete network elements graphically. With these capabilities, modeling becomes a simple point and click exercise. The on-line tutorials have step-by-step instructions for performing common tasks in the graphical editor, and Lesson 1 also offers assistance.

### 5.2.2 Working with Network Elements Within the Graphical Editor

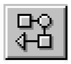
Most network editing tasks can be performed using only your mouse. The pull-down menus and AutoCAD command line also offer the ability to perform many of these tasks, but by simply pointing and clicking with the mouse you will be able to:

- Create New Elements
- Select Elements
- Edit Elements
- Move Elements
- Delete Elements
- Annotate the Drawing

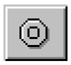
**As you place your mouse over each element, a tool-tip is diplayed informing you of the element's label and annotations.**

### 5.2.3 Creating New Elements

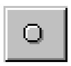
The tool palette contains all of the tools for adding network elements to the drawing. These element tools include:



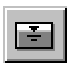
**Pipe Layout Tool** - Pipes connect the other elements to form the sewer network. The pipes are the conveyance elements that carry flow through the network to its eventual discharge point at an outlet. The pipe tool creates either gravity pipes (represented by two parallel lines) or pressure pipes (represented by a single line) depending on the pipe's location within the sewer network.



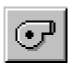
**Manhole Tool** - Manholes are locations where loads enter the gravity portion of the sewer network.



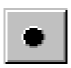
**Junction Chamber Tool** - Junction chambers are locations where upstream flows in a gravity system combine. No loads enter the sewer at these points.



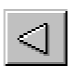
**Wet Well Tool** - Wet wells represent boundary conditions between pressure and gravity portions of a SewerCAD network. They serve as collection points for gravity systems, and as a HGL boundary node for the pressure system. Dry loads can also enter the sewer network at these locations.



**Pump Tool** - Pumps are used to add energy to the system to overcome elevation differences and headlosses.



**Pressure Junction Tool** - Pressure junctions are connections between two or more pressure pipes of varying characteristics. Loads may enter a pressure portion of a network through a pressure junction.



**Outlet Tool** - Outlets represent ultimate termination points in a sanitary sewer network.

Although the elements can all be inserted individually, the most rapid method of network creation is through the **Pipe Layout** tool. The **Pipe Layout** tool enables you to connect existing nodes with new pipes, and also allows you to create new nodes as you lay out the pipes.

For example, when the **Pipe Layout** tool is active, clicking within the drawing pane will insert a node. Clicking again at another location will insert another node and connect a pipe between them. Use the on-line tutorials to experience it interactively.

### 5.2.4 Changing the Pipe Layout Tool to Insert a Different Type of Node

While laying out a network, you may need to change the type of node that the **Pipe Layout** tool inserts. This can be done very easily by following the steps outlined below:

With the **Pipe Layout** tool active, right-click in the drawing pane.

1. A pop-up menu will appear with a list of available element types.
2. Select an element type from the pop-up menu.

**The cursor appearance will change to reflect the type of node to be inserted.**

### 5.2.5 Morphing Elements

Occasionally, you may find that you need to replace a node with a different type of node. You can make this change through a process called morphing.

Morphing enables you to change an existing network node type, without having to delete and re-create the node and all of its connecting links. Information types that are common between the existing and new elements will be copied into the new element. To morph an existing element into a different type of element:

1. Select the new element type from the **Tool Palette**.
2. In the drawing pane, place the cursor over the old element and click.
3. You will be prompted to verify that you want to morph. Answer yes to perform the morph, or answer no and a new element will be added at the specified location. If you accidentally morph an element, this action can be undone by selecting **Edit / Undo** from the pull-down menus.

### 5.2.6 Splitting Pipes

You may encounter a situation in which you need to add a new node in the middle of an existing pipe. For example, you may want to insert a new inlet to capture excessive surface flow in StormCAD, a new junction to represent additional demand in WaterCAD, or a new manhole to maintain maximum access hole spacing in SewerCAD.

You can split existing pipes simply by inserting a node along the pipe as follows:

From the **Tool Palette**, select the node type.

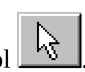
1. In the drawing pane, place the cursor over the pipe and click
2. You will be prompted to confirm that you wish to split the pipe. If you choose to split the pipe, the node will be inserted and two new pipes will be created with the same characteristics as the original pipe (lengths are split proportionally).
3. If you choose not to split the pipe, the new element will be placed on top of the pipe without connecting to anything.

If you accidentally split a pipe, this action can be undone by selecting **Edit / Undo** from the pull-down menus.

### 5.2.7 Selecting Elements

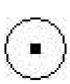
You can select one element or a group of elements from drawing pane on which to perform various operations such as moving, deleting, and editing.

#### Selecting Elements (Stand-Alone Mode)

1. In Stand-Alone model activate the **Select** tool .
2. To select a single element, simply click the desired element. To select a group of elements, click the drawing pane and drag the mouse to form a selection box around the elements you want to select, then click again to choose the other corner of the selection box. All elements that are fully enclosed within the selection box will be selected.

To toggle the selected status of one or more elements, you can follow the same instructions as above while holding down the **Shift** key. There are also additional ways to select elements through the **Edit** menu.

When an element is selected in the Stand-Alone drawing pane, it will be displayed with at least one grip. A grip is a black box, as shown below, that indicates the figure's insertion point. The label of a selected item, or the number of selected items, will be displayed in the status bar.

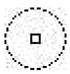


### Selecting Elements (AutoCAD Mode)

Within AutoCAD, the **Select** tool does not need to be active when making a selection. Simply use the standard AutoCAD selection techniques.

AutoCAD also offers a variety of other selection methods that are outlined in the AutoCAD documentation.

When an element is selected in AutoCAD, it may be displayed in a dashed linetype, and the grips may become visible, as shown below. The exact display depends on how the element was selected and the value of the AutoCAD variable GRIPS.



### Selecting Upstream / Downstream Elements

You can select all the elements upstream or downstream of a selected gravity node by right clicking on the node and selecting **Select / Elements Upstream / Downstream from …** from the pop-up menu. If in AutoCAD, make sure that **Right Click Context Menu** is checked in the **Global Options** dialog.

## 5.2.8 Editing Elements

There are several methods for editing network element data, including Database Connections, FlexTables, and the **Alternative Manager**.

Perhaps the most common method of changing element data, however, is from an individual element's editor dialog. To edit a single element, use the **Select** tool in Stand-Alone mode or in AutoCAD mode.

In Stand-Alone mode and in AutoCAD 2000i, editing a single element is very easy. Simply double-click the element, and the **Editor** dialog will open. Alternatively, you can right-click the element and select **Edit…** from the pop-up menu.


In AutoCAD 2000 and AutoCAD R14, the process is slightly different. First, click the **Select** tool, then click the element you wish to edit. If you are using AutoCAD 2000, or you are using AutoCAD R14 and you have **Right Click Context Menu** checked in the **Global Options** dialog, you can also right-click to activate the pop-up context menu.

Right-click context menus can provide easy access to common functions and actions.

In AutoCAD 2000i, like in Stand-Alone, you can also double-click the element to bring up its editor.

## 5.2.9 Moving Elements

You can change the location of elements easily, whether you are in Stand-Alone or AutoCAD mode.

The first step is to select the element(s) to be moved. Next, click to drag the element, and release to drop it at its new location. In AutoCAD mode, you can accomplish this by dragging the grips. When a node is moved to a new location, all connected pipes will remain attached, and pipe lengths will automatically update (unless the pipe has a user-defined length or you are working in schematic mode).

> **In AutoCAD, this command is the equivalent of the STRETCH command, not the MOVE command. There are also several other methods of moving items within AutoCAD. For more information regarding moving elements within AutoCAD, please refer to your Autodesk documentation.**

In the same fashion, you can graphically change the location of element labels and annotation relative to the element.

A node element can also be moved by editing its coordinates in the element's editor, in FlexTables, or through database connections.

### 5.2.10 Deleting Elements

Deleting elements is quite easy. Simply select the element(s) to be deleted, and press the **Delete** key on the keyboard. Note that the integrity of the network is automatically maintained when deletions are performed. This means that when a node is deleted, any connecting pipes are also deleted to prevent "dangling" pipes that would cause the network to be invalid.

There are also several other methods of deleting elements, including selecting **Edit / Delete** from the pull-down menus, or typing **ERASE** at AutoCAD's command line.

### 5.2.11 Other Tools

Although this product is primarily a modeling application, some additional drafting tools can be helpful for intermediate calculations and drawing annotation. AutoCAD, of course, provides a tremendous number of drafting tools.

In Stand-Alone mode, drafting and annotation tools allow you to add polylines (multi-segmented lines), rectangles, and text to the drawing pane.

#### Line-Enclosed Area

In Stand-Alone mode, you can calculate the enclosed area of any closed polyline. This feature can be especially helpful for determining the size of storm catchments or land-use areas.

Simply right-click the closed polyline, and select **Enclosed Area** from the pop-up menu. The **Area** dialog will open, displaying the calculated area of the polyline enclosure.

> **This tool is only available for closed polylines. To close an open-ended polyline, right-click it and select Close from the pop-up menu.**

Although this feature is not provided in AutoCAD mode, you can determine the area of any AutoCAD polyline by performing a **LIST** command.

## 5.3 Selection Sets

Selection sets are user-defined groups of network elements. They allow you to predefine a group of network elements that you want to manipulate together. Selection sets are defined through the **Selection Set Manager** by selecting **Tools / Selection Sets** from the pull-down menus.

### 5.3.1 Selection Set Manager

- **Add -** Add a new selection set.
- **Edit -** Edit an existing selection set.
- **Duplicate -** Copy an existing selection set.
- **Delete -** Delete an existing selection set.
- **Rename** - Rename an existing selection set.
- **Notes -** Add a note regarding the selection set.

### 5.3.2 New Selection Set

After clicking **Add** in the **Selection Set Manager**, a dialog appears. Simply enter the name of your new selection set in the dialog. Click **OK** to name the selection set, or **Cancel** to exit the dialog without creating a selection set.

### 5.3.3 Selection Set Dialog

In this dialog, you will notice two panes. A listing of all the elements in the network is displayed in the **Available Items** pane. To add items to the **Selected Items** pane, select the desired elements in the available list and click the **[>]** button under **Add**. To add all the elements to your selection set, click the **[>>]** button.

Additionally, you can use the **Select** button to highlight items in the **Available Items** pane using a variety of powerful selection techniques, or by graphically selecting elements from the drawing. It will also allow you to invert the selection set, thereby unselecting the ones already selected and selecting the ones not already selected. You can also clear the selected items using the **Select** button.

The features mentioned above are also available to remove items from the **Selected Items** pane.

### 5.3.4 Duplicate Selection Set

Click **Duplicate** make a copy of the highlighted selection set.

### 5.3.5 Delete Selection Set

Click **Delete** to delete the highlighted selection set.

### 5.3.6 Rename Selection Set

Click **Rename** to open a dialog that allows you to change the name of the highlighted selection set.

### 5.3.7 Selection Set Notes

Click **Notes** to input free form paragraph text that will be associated with the highlighted selection set.

## 5.4 Find Element

This is a powerful feature that allows you to quickly locate any element in the drawing by its label. It performs a case *insensitive* search. The **Find Element** feature is available from the **Edit** menu on the main window.

To find an element:

Choose **Edit / Find Element** from the pull-down menus.

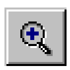
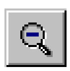
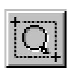
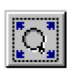
1. Type the label of the element you wish to find, or click the list box to choose from a sorted list of elements in the system.
2. You may wish to choose a Zoom Factor from the list provided. 100% is the default Zoom Factor. If you wish to magnify the view of the drawing, then choose a Zoom Factor greater than 100%. To decrease the view of the drawing, choose a Zoom Factor less than 100%.
3. Click **OK**.

# 5.5 Zooming

Zooming controls how large or small a drawing appears on the screen. Zooming is helpful when you want to enlarge the display to see the drawing's details, or to reduce the display to see an entire drawing. Zooming does not change the actual size of the drawing, only the size of the current view.

You can zoom by doing one of the following:

From the **View** pull-down menu or the toolbars you can perform the following zoom operations:

| | | |
|---|---|---|
| Zoom In |  | Enlarge the view of the drawing. |
| Zoom Out |  | Decrease the view of the drawing. |
| Zoom Window |  | Choose the portion of the drawing to fit in the window by drawing a selection box around it. |
| Zoom Extents |  | Bring all elements in the drawing into view. |
| Zoom Previous |  | Return to the most recent view of the drawing. |
| Zoom Center | | Center the location of specific coordinates within the drawing pane. |

**You can use the Plus key (+) and the Minus key (-) on the numeric keyboard as a shortcut for zooming in and out respectively (Stand-Alone mode only).**

**You can also zoom in and out by holding down the ctrl key and using the mouse wheel.**

## 5.5.1 Zoom Center

The **Zoom Center** dialog provides you with a quick way to zoom to any area of your drawing. This feature is useful if you want to start laying out a network around certain coordinates, or if you know the coordinates of an existing element that you would like to locate.

To use Zoom Center:

1. Select **View / Zoom Center** from the pull-down menus.

2. In the **Zoom Center** dialog, enter the coordinates to which you would like to zoom.
3. Select a zoom factor if you would like to increase or decrease the magnification.
4. Click **OK,** and the specified coordinates will be located at the center of the drawing.

### 5.5.2 Aerial View

The Aerial View is a small navigation window that provides a graphical overview of your entire drawing. You can toggle the **Aerial View** window on or off by selecting **View / Aerial View** from the pull-down menu.

A Navigation Rectangle is displayed in the **Aerial View** window. This Navigation Rectangle provides a "you are here" indicator showing you current zoom location with respect to the overall drawing. As you pan and zoom around the drawing, the Navigation Rectangle will automatically update to reflect your current location.

You can also use the **Aerial View** window to navigate around your drawing. To pan, simply click the Navigation Rectangle to drag it to a new location. To zoom, click anywhere in the window to specify the first corner of the Navigation Rectangle, and click again to specify the second corner.

#### *In AutoCAD mode:*

Refer to the AutoCAD on-line Help for a detailed explanation.

#### *In Stand-Alone mode:*

With **Aerial View** window enabled (by selecting the **View / Aerial View** from the pull-down menu), click and drag to draw a rectangular view box in the aerial view. The area inside this view box is displayed in the main drawing window. Alternately, any zooming or panning action performed directly in the main window updates the size and location of the view box in the **Aerial View** window.

The **Aerial View** window contains the following buttons:

- **Zoom Extents** - Display the entire drawing in the **Aerial View** window.
- **Zoom In** - Decrease the area displayed in the **Aerial View** window.
- **Zoom Out** - Increase the area displayed in the **Aerial View** window.

To resize the view box directly from the **Aerial View** window, simply draw a new rectangular view box. To change the location of the view box directly in the **Aerial View** window, you can either drag the view box frame or create a new one.

## 5.6 Drawing Review

The **Drawing Review** window allows you to quickly navigate to and review any group of elements. This tool is particularly useful for finding potential problems in a network. These problems may result from data entry errors or data discrepancies in the source (database, Shapefile, or CAD drawing) from which a model was imported.

By default, when the **Drawing Review** window opens, all elements will appear in the list. You can work with any subset of elements by choosing one of the following items:

- **Select / Custom** - Allows you to choose any set of elements to review using the **Selection Set** dialog.
- **Select / All Elements** - Automatically selects all available elements.
- **Select / Nodes in Close Proximity** - Allows you to select all nodes that are within a user-defined tolerance of another node. The tolerance is defined in the **Nodes in Close Proximity** dialog, which opens when this option is selected. This tool is useful for finding and correcting connectivity problems.

For example, if two nodes are close too each other the may actually be the same node, and one of them needs to be deleted.

- **Select / Pipe-Split Candidates** - Allows you to find nodes that are closer to a pipe than a user-defined tolerance, but are not connected to the system. The tolerance is defined in the **Pipe-Split Candidates** dialog, which opens when this option is selected. This option is useful for finding and correcting connectivity problems.
- **Select / Orphaned Nodes** - Allows you to select all orphaned nodes in your network. A node is an orphan when it is not connected to any pipe.
- **Select / Elements with Messages** - Allows you to select all the elements that have warnings or error messages, appearing in the **Messages** tab of an **Element Editor** dialog. This is useful for correcting data entry errors.
- **Select / Clear Drawing Review Messages** - Allows you to reset Drawing Review messages for all elements in the list. Drawing Review messages are automatically added during various Import operations such as Polyline to Pipe Import and Land Development Desktop Import (SewerCAD or StormCAD only). After you review and fix these problems, you may want to clear the review messages. If you want to retain some of the drawing review messages, simply remove those elements from the list prior to performing this operation.

The elements you select will appear in the primary list located along the left side of the **Drawing Review** window.

- **Go To** - To navigate to an element, select the desired element in the list and press the **Go To** button.
- **Next / Prev** - To navigate to the elements sequentially, use the **Next** or **Prev** buttons.
- **Zoom** - You can control the degree to which the drawing review zooms into the selected element by choosing a zoom factor from the field labeled **Zoom**, located in the lower right corner of the dialog.

**You can double-click an element in the list to quickly navigate to that element.**

**If you know the name of the element to which you wish to navigate, type the label in the field located above the element list and click the Go To button.**

**All menus and toolbars will remain available even when the Drawing Review window is open. This allows you to navigate to and fix any problems that you find.**

Use the **Drawing Review** window in conjunction with the **QuickView** window to review the data for the selected elements.

To access the **Drawing Review** dialog, select **Edit / Review Drawing** from the pull-down menus.

### 5.6.1 Selection Tolerance

Some *select* operations require you to specify a tolerance for defining which nodes will be selected for the **Drawing Review** window.

- **Elements in Close proximity** - If the distance between the elements in the drawing is within the specified tolerance, those elements will be selected for display in the **Drawing Review** window.
- **Pipe Split Candidates** - If the distance between a node and a pipe is within the specified tolerance, it will be selected for display in the **Drawing Review** window.

# 5.7 Relabel Elements

Element relabeling allows you to modify the labels of a selected set of elements. This feature is especially useful with a model built from a database that uses numeric IDs to identify elements, making it difficult to distinguish between the different types of elements in the system. With element relabeling, you can quickly append a prefix such as ‘P-’ to all pipes in your system so that it is obvious which labels belong to elements representing pipes.

The **Relabel Elements** dialog contains two sections:

- **Relabel Operations** - Allows you to select and define the operations you want to perform.
- **Elements Selected** - Allows you to select which elements in your project you want to relabel.

To access the **Relabel Elements** dialog, select **Tools/Relabel Elements** from the pull-down menu.

## 5.7.1 Relabel Operations

The element relabeling tool allows you to perform three types of operations on a set of element labels: Replace, Renumber, and Append. The active relabel operation is chosen from the list box in the **Relabel Operations** section of the **Relabel Elements** dialog. The entry fields for entering the information appropriate for the active relabel operation appear below the **Relabel Operations** section. The following list presents a description of the available element relabel operations.

- **Replace** - This operation allows you to replace all instances of a character or series of characters in the selected element labels with another piece of text. For instance, if you selected elements with labels P-1, P-2, P-12, and J-5, you could replace all the P's with the word Pipe by entering ‘P’ in the **Find** field, ‘Pipe’ in the **Replace With** field, and clicking the **Apply** button. The resulting labels are Pipe-1, Pipe-2, Pipe-12, and J-5. You can also use this operation to delete portions of a label. Suppose you now want to go back to the original labels. You can enter ‘ipe’ in the **Find** field and leave the **Replace With** field blank to reproduce the labels P-1, P-2, P-12, and J-5. There is also the option to match the case of the characters when searching for the characters to replace. This option can be activated by checking the box next to the **Match Case** field.
- **Renumber** - This operation allows you to generate a new label, including suffix, prefix, and ID number for each selected element. For example, if you had the labels P-1, P-4, P-10, and Pipe-12, you could use this feature to renumber the elements in increments of five, starting at five, with a minimum number of two digits for the ID number field. You could specify a prefix ‘P-’ and a suffix ‘-Z1’ in the **Prefix** and **Suffix** fields, respectively. The prefix and suffix are appended to the front and back of the automatically generated ID number. The value of the new ID for the first element to be relabeled, 5, is entered in the **Next** field. The value by which the numeric base of each consecutive element is incremented, 5, is entered in the **Increment** field. The minimum number of digits in the ID number, 2, is entered in the **Digits** field. If the number of digits in the ID number is less then this value, zeros are placed in front of it. Click the **Apply** button to produce the following labels: P-05-Z1, P-10-Z1, P-15-Z1, and P-20-Z1.
- **Append** - This operation allows you to append a prefix, suffix, or both to the selected element labels. Suppose that you have selected the labels 5, 10, 15, and 20, and you wish to signify that these elements are actually pipes in Zone 1 of your system. You can use the append operation to add an appropriate prefix and suffix, such as ‘P-’ and ‘-Z1’, by specifying these values in the **Prefix** and **Suffix** fields and clicking the **Apply** button. Performing this operation yields the labels P-5-Z1, P-10-Z1, P-15-Z1 and P-20-Z1. You can append only a prefix or suffix by leaving the other entry field empty. However, for the operation to be valid, one of the entry fields must be filled in.

The selection set of elements on which the relabel operation is to be performed can be selected in the **Elements** section of the **Relabel Elements** dialog.

To access the **Relabel Elements** dialog, select **Tools / Relabel Elements** from the pull-down menus.

### 5.7.2 Elements Selected

The **Elements** section contains a pane that lists the elements to be relabeled. You can select the set of elements that appears in this pane by clicking the **Select** button. This accesses the **Selection Set** dialog, where you can pick a set of elements from all the elements currently in the project.

For the **Append** and **Replace** operations, the order that the elements appear in the text pane does not affect the results of the operation. However, for the **Renumber** operation, the order in which the elements appear in the text pane determines the order in which they will be renumbered. The default order in which the elements appear in the text pane is in the alphanumeric order of the element labels, called Ascending order. If you wish to change this order, click the **Sort** button, and select **Network Order** to put the elements in the order they appear in the network, **Descending Order** to put them in reverse alphanumeric order, or **Ascending Order** to put them back in alphanumeric order.

## 5.8 Element Labeling

The **Element Labeling** dialog is used to specify the automatic numbering format of new elements as they are added to the network. The following options are available:

- **Element** - View the type of element to which the label applies.
- **Next** - Enter the integer you want to use as the starting value for the ID number portion of the label. The program will generate labels beginning with this number, and will choose the first available unique label.
- **Increment** - Enter the integer that will be added to the ID number after each element is created to yield the number for the next element.
- **Prefix** - Enter the letters or numbers that will appear in front of the ID number for the elements in your network.
- **Digits** - Enter the minimum number of digits that the ID number will have. For instance, 1, 10, and 100 with a digit setting of two would be 01, 10, and 100.
- **Suffix** - Enter the letters or numbers that will appear after the ID number for the elements in your network.
- **Preview** - View an example of what the label will look like based on the information you have entered in the previous fields.

Changes to the element labeling specifications will only affect the numbering of **new** elements. Existing elements will not be affected. In order to adjust the numbering of existing elements, utilize the **Relabel Elements** option accessible from the **Tools** menu.

> **Pipe labeling can be aligned with the pipes or be displayed horizontally, depending on the Pipe Text setting specified in the Drawing Options dialog.**

You can control the angle at which the text flips from one side of the pipe to the other to read in the opposite direction, when the pipe direction on a plot is nearly vertical. By default, the text flips direction when the pipe direction is 1.5 degrees, measured counter-clockwise from the vertical. You may modify this value by inserting a **TextFlipAngle** variable in the Haestad.ini file that is located in the program file of your Haestad directory, and specifying the angle at which the text should flip. The angle is measured in degrees, counter-clockwise from the vertical. For instance, if you want the text to flip when the pipe direction is vertical, you should add the following line to the Haestad.ini file:

TextFlipAngle=0.0

Reasonable values typically fall in the range 15.0 deg to -15.0 deg.

To access the **Element Labeling** dialog, select **Tools / Element Labeling** from the pull-down menus.

## 5.9 Quick View

The **Quick View** window provides you with a fast way to edit or view the data associated with any element in the network without having to open the element dialog. It is a floating window that includes input and output information for any element that you have selected. It also includes a convenient color-coding legend. Three tabs are provided on the window:

- **Input** - Contains input data for the selected element.
- **Output** - Contains output data for the selected element.
- **Legend** - Displays ranges of the active color-coding.

When the **Quick View** window is open, the data for an entity will immediately be displayed when you select it within the graphical editor. Once an element has been selected, click on any editable field on the **Input** tab to edit the associated value. Edits will be committed when you leave the **Quick View** window. Changes made through the **Quick View** window can be undone/redone by accessing the **Edit** menu.

**You can change the size of the Label, Value, and Unit columns on the Input/Output tabs by using the resizing bar at the top of the Quick View window.**

**You can highlight an Input or Output attribute (e.g. Demand), by clicking the label of that attribute in the Quick View window. This highlighting provides for better visual feedback, for example, when monitoring the pressures at several nodes.**

The **Quick View** window can be accessed by clicking the **Quick View** window button  on the toolbar. You can also select **View / Quick View** from the pull-down menus.

# Chapter 6

# Hydraulic Element Editors

## 6.1 Overview

The primary component of a SewerCAD project is the collection system model. A collection system model may contain multiple independent (not connected) networks in a single project file. Each network may contain gravity elements, and pressure elements, in which the flow is under pressure due to the presence of a pumping station upstream. The element types that are used to form a network are:

- **Manholes** - These are node elements used to model access hole structures. At a manhole, you can enter a local load and also model the headloss associated with the structure.
- **Junction Chambers** - These are node elements used to model underground structures. Unlike a manhole, no local load can be entered at a junction chamber; however, junction headloss associated with this structure can be modeled.
- **Wet Wells** - These are storage nodes typically used in conjunction with one or more pumps to model a pumping station. Wet wells can be defined with either a constant section area or variable section area. A local loading may also be added at a wet well.
- **Pumps** - Pumps are elements that add head to the system as water passes through them. A pump is typically defined by a pump curve and control elevations, at which the pump turns off or on.
- **Pressure Junctions** - These are node elements used to model a junction under pressure at the downstream end of one or several pressure pipes.
- **Outlets** - These are node elements that define the "root," or most downstream element of a SewerCAD network. They specify the starting hydraulic grade line for the backwater analysis. Gravity networks contain only one outlet element. However, a pump may pump into more than one forcemain, thus allowing split flow and possibly more than one outlet in pressure networks.
- **Gravity Pipes** - These are link elements of constant shape, material, size, and slope that are used to transport the discharges from node to node.
- **Pressure Pipes** - These are link elements of circular shape and of constant material and size used to transport the discharges under pressure from node to node.

This chapter presents a detailed look at the input and output data used in a SewerCAD project, and the way it is organized in the graphical user interface. First, a description of the elements used to model the sewer collection system is provided. Second, the various means of entering loading data are described. Then, prototypes are discussed as a way to initialize new model elements with default values. Finally, the user's ability to augment SewerCAD's existing attribute set with User-Data extensions are discussed.

> **A collection system model will not be considered valid for calculation if the number of pipes exceeds the licensed size. To determine how many pipes you are licensed for, select the Help / About SewerCAD from the pull-down menu. Click the Registration button. A single SewerCAD project file can contain any number of valid networks, but if the total number of pipes exceeds the licensed size, the project will not calculate.**

# 6.2 Element Editors

## 6.2.1 Using Element Editors

The **Element Editors** allow you to edit all input data and view all output data defining a single network element.

> **Element data may also be viewed/edited more efficiently through FlexTables, which display all the data in customizable tabular format, allowing you to perform functions such as sorting, filtering, and global editing. The data may also be quickly reviewed through the Quick View window.**

You can move to connecting elements from the selected element's editor by using the following buttons.

Click this button to move to the default upstream element of the selected gravity element or to the default-connected element of the selected pressure element.

Click this button to move to the default downstream element of the selected gravity element.

Click the side-triangle button to open a pop-up menu to select one of the available connecting elements.

To access an **Element Editor**:

*Stand-Alone:* *Double-click the element you wish to edit, or right-click the element and select* **Edit** *from the drop-down menu.*

*AutoCAD R14:* *Pick the* **Select** *tool and click the element you wish to edit. If the* **Right-Click Context Menu** *option is enabled, you can also right-click the element and select* **Edit** *from the drop-down menu.*

*AutoCAD 2000:* *Pick the* **Select** *tool and click the element you wish to edit, or select the element and choose* **Edit** *from the drop-down menu.*

*AutoCAD 2000i:* *Double-click the element you wish to edit, or right-click the element and select* **Edit** *from the drop-down menu.*

## 6.2.2 Manholes

Manholes are the elements used to model the access holes in a sewer collection system. The **Manhole Editor** organizes the related input data and calculated results into the following tabs:

- **General -** General manhole information containing geographical data and hydraulic results.
- **Headlosses -** Headloss calculation method and parameters.
- **Diversion -** Diversion target, rating table, and results.
- **Loading -** Loading data, which is composed of sanitary loads, wet weather loads, and known flows.
- **Design -** Constraints used during automatic design.
- **Cost -** Cost analysis input/output data used when performing cost analysis calculations.
- **User Data -** Additional user-entered data. For instance, you can add new fields such as the manhole installation date.
- **Messages -** Calculation messages, such as computation warnings or error messages, and user-entered notes and descriptions.

For further details, refer to the topics describing each tab.

### 6.2.3 Junction Chambers

Junction chambers are used to model underground nodal structures in gravity sewer collection systems. Unlike a manhole, no local load can be entered at a junction chamber. Headloss associated with junction chambers can also be modeled. The **Junction Chamber Editor** organizes the related input data and calculated results into the following tabs:

- **General -** General junction chamber information containing geographical data and hydraulic results.
- **Headlosses -** Headloss calculation method and parameters.
- **Diversion -** Diversion target, rating table, and results.
- **Design -** Constraints used during automatic design.
- **Cost -** Cost analysis input/output data used when performing cost analysis calculations.
- **User Data -** Additional user-entered data. For instance, you can add new fields such as the junction installation date.
- **Messages -** Calculation messages, such as computation warnings or error messages, and user-entered notes and descriptions.

### 6.2.4 Wet Wells

Wet wells are used to model the storage structures in sewer collection systems. They are storage nodes usually used in conjunction with one or several pumps to model a pumping station. Wet wells can be defined with either a constant section area or a variable section area. A local loading may also be added at a wet well. The **Wet Well Editor** organizes the related input data and calculated results into the following tabs:

- **General -** General wet well information containing geographical data and hydraulic results.
- **Section -** Geometric characteristics of the well, as well as water level limits.
- **Loading -** Load data, which is composed of sanitary loads, wet weather loads, and known flows.
- **Cost -** Cost analysis input/output data used when performing cost analysis calculations.
- **User Data -** Additional user-entered data. For instance, you can add new fields such as the well construction date.
- **Messages -** Calculation messages, such as computation warnings or error messages, and user-entered notes and descriptions.

### 6.2.5 Pumps

Pumps are used to model devices that add energy to sewer systems as water passes through them. A pump is typically defined by a pump curve and control elevations, at which the pump turns off or on. A pump is conceptually a composite element, modeled as a link with junction nodes on either end, although it is represented in the graphical editor by a single icon. The **Pump Editor** organizes the related input data and calculated results into the following tabs:

- **General -** General pump information containing geographical data, the pump curve data, the initial setting, and the hydraulic results.
- **Controls -** Data specifying the on/off elevation settings of the pump, as well as relative speed factor settings in the case of a variable speed pump.
- **Cost -** Cost analysis input/output data used when performing cost analysis calculations.
- **User Data -** Additional user-entered data. For instance, you can add new fields such as the pump installation date.

- **Messages -** Calculation messages, such as computation warnings or error messages, and user-entered notes and descriptions.

### 6.2.6 Pressure Junctions

Junction structures are used to model junctions under pressure at the downstream end of one or several pressure pipes. The **Pressure Junction Editor** organizes the related input data and calculated results into the following tabs:

- **General -** General manhole information containing geographical data and hydraulic results.
- **Loading -** Load data, which is composed of sanitary loads, wet weather loads, and known flows.
- **Cost -** Cost analysis input/output data used when performing cost analysis calculations.
- **User Data -** Additional user-entered data. For instance, you can add new fields such as an observed pressure at the junction.
- **Messages -** Calculation messages, such as computation warnings or error messages, and user-entered notes and descriptions.

### 6.2.7 Outlets

Outlets are the node elements that define the "root," or most downstream element of a SewerCAD network, and specify the starting hydraulic grade line for the backwater analysis. The **Outlet Editor** organizes the related input data and calculated results into the following tabs:

- **General** - General outlet information containing geographical data and hydraulic results.
- **Design** - Constraints used during automatic design.
- **Cost -** Cost analysis input/output data used when performing cost analysis calculations.
- **User Data** - Additional user-entered data. For instance, you can add new fields such as the outlet installation date.
- **Messages** - Calculation messages, such as computation warnings or error messages, and user-entered notes and descriptions.

### 6.2.8 Gravity Pipes

Gravity pipes are used to model the pipes in the system in which the flow is discharging downstream due to gravity. The **Gravity Pipe Editor** organizes the related input data and calculated results into the following tabs:

- **General** - General pipe information containing physical characteristics data and hydraulic results.
- **Profile** - Information regarding the pipe's physical and hydraulic profile.
- **Design** - Constraints used during automatic design.
- **Infiltration** - Infiltration data, which may be proportional to the pipe characteristics or defined as a lump sum.
- **Cost -** Cost analysis input/output data used when performing cost analysis calculations.
- **User Data** - Additional user-entered data. For instance, you can add new fields such as the pipe installation date.
- **Messages** - Calculation messages, such as computation warnings or error messages, and user-entered notes and descriptions.

### 6.2.9 Pressure Pipes

Pressure pipes are used to model the pipes carrying flow under pressure, discharging from a pumping station located upstream in the sewer collection system. The **Pressure Pipe Editor** organizes the related input data and calculated results into the following tabs:

- **General** - General pipe information containing dimension and physical characteristics data, and hydraulic results.
- **Controls** - Control data used to specify whether the pipe is open or closed based on the HGL or pressure at any given node in the system.
- **Cost** - Cost analysis input/output data used when performing cost analysis calculations.
- **User Data** - Additional user-entered data. For instance, you can add new fields such as the pipe installation date.
- **Messages** - Calculation messages, such as computation warnings or error messages, and user-entered notes and descriptions.

## 6.3 Element Editors' Tabs

### 6.3.1 General Tab

#### Manholes, Junction Chambers, and Outlets

*General Tab (for Manholes, Junction Chambers, and Outlets)*

The **General** tab for manholes, junction chambers, and outlets is organized into the following sections:

- **General** - General data about the node.
- **Structure** - Geometric and elevation data.
- **Hydraulic Summary** - Hydraulic results for manholes or junction chambers.
- **Tailwater Hydraulics** - Tailwater data for an outlet.
- **Flow Summary** - Total flow and distribution of the flow at a node.

For further details, refer to the topics describing each section.

*General Section*

The **General** section allows you to enter general information about the node, such as:

- **Label** - Unique name by which an element will be referred to in reports, error messages, and tables.
- **X (Easting)** - The location of the node may be represented as an X-value or an Easting value, depending on individual preferences.
- **Y (Northing) -** The location of the node may be represented as a Y-value or a Northing value, depending on individual preferences.
- **Ground Elevation -** Elevation of the ground surface at the node.
- **Station / Calculated Station** - Distance along the alignment of pipes. The starting station can be specified at an outlet or wet well, and is calculated for nodes and junctions.

**The Station attribute is only editable for Outlet and Wet Well elements. For all other elements, the station is computed during a calculation based on the initial station and pipe lengths.**

### *Structure Section*

The **Structure** section allows you to enter data pertaining to the node's structure:

- **Bolted Cover -** Specify whether a manhole cover is bolted, in which case the flow is contained inside the structure when the water level rises to the rim elevation, instead of spilling over.
- **Set Rim to Ground Elevation -** This check box enables or disables a data entry shortcut. If the box is checked, the manhole or outlet rim elevation will be set equal to the ground elevation automatically. During an automatic design, the structure rim elevations will be the same as the ground elevation.
- **Rim Elevation -** This field is editable only when the **Set Rim to Ground Elevation** check box for a manhole or outlet is unchecked.
- **Sump Elevation -** Elevation of the bottom of a manhole or outlet.
- **Structure Diameter -** Cross-sectional diameter of a manhole or junction chamber. This value is used in hydraulic calculations and profile drawings, but not in plan view.
- **Top Elevation -** Elevation of the top of a junction chamber.
- **Bottom Elevation -** Elevation of the bottom of a junction chamber.

**A warning will be posted to the calculation log whenever the defined sump elevation is invalid (i.e. above any connecting pipe invert).**

**During automatic design calculations, the program automatically sets sump inverts according to the design constraints specified in the active Design Constraints Alternative.**

### *Hydraulic Summary Section*

The **Hydraulic Summary** section, available on the **General** tab of the **Manhole** and **Junction Chamber Editors**, provides the following quick summary of the calculated hydraulic results:

- **Hydraulic Grade Line In** - Hydraulic grade at the downstream end of the incoming pipe section.
- **Gravity Element Headloss** - Headlosses associated with factors such as mixing and change of direction.
- **Hydraulic Grade Line Out** - Hydraulic grade at the upstream end of the outgoing pipe section.

### *Tailwater Hydraulics Section*

The **Tailwater Hydraulics** section, available on the **General** tab of the **Outlet Editor**, contains the following fields:

- **Tailwater Condition** - Select the outfall condition (crown, free-outfall, or user-specified) from the choice list. The program performs a backwater analysis throughout the sewer system starting from the outfall condition selected.
- **Tailwater Elevation** - Only editable under user-specified tailwater conditions. For all other tailwater conditions, the field is calculated.
- **Hydraulic Grade Line Out** - The computed tailwater elevation.

**You may want to let the program compute an ideal outfall invert elevation and profile as a preliminary design. To do this, simply run an automatic design and let the program compute a good estimate based on cover requirements, slope constraints, and pipe sizes.**

### *Flow Summary Section*

This section reports a summary of the total flow at a node and the distribution of this flow.

For **Steady State** mode the fields displayed are as follows:

- **Total Dry Weather Flow** - Total flow at a node resulting from sewage generated during dry weather from the network upstream of a given node.
- **Total Wet Weather Flow** - Total flow at a node resulting from the intrusion of rainfall water into the sewer system from the upstream network. Wet weather load consists of groundwater infiltration and rainfall inflow. Groundwater infiltration occurs in gravity pipes while inflows occur at manholes and wet wells.
- **Pumped Flow** - Portion of the total flow that comes from pumps located in the upstream network, when the **Use Pumped Load** toggle in the **Calculation Options** dialog is On. In this case, the flow coming out of pumps is treated as a fixed flow, to which peaking factors do not apply.
- **System Known Flow** - Portion of the total flow derived from manually entered Known Flows upstream of the reporting point. Known flows are not additive except at network junctions. The Known Flow component will remain constant until it encounters a downstream Known Flow of a different value.
- **Total Flow** - Total flow going through an element of the network.

For **EPS** mode the fields displayed are as follows::

- **Total Flow -** Represents the total flow coming to the structure, which includes the upstream flow through pipes, incoming diverted flow, and the local sanitary loads and inflows.
- **Diverted Flow** - Represents the amount of the total flow, which is diverted from the inlet to the diversion target.
- **Flow In -** Represents the portion of the flow established locally at the selected node, such as unit dry weather and wet weather loads.

**You can toggle between Steady State and EPS modes by clicking the GO button and changing the Calculation Type.**

## Wet Wells

### *Wet Well General Tab*

The **General** tab for wet wells is organized into the following groups:

- **General -** General data about the node.
- **Hydraulic Summary -** Reports calculated hydraulic grade line leaving the well.
- **Flow Summary -** Total flow and distribution of the flow at the wet well.

For further details, refer to the topics describing each section.

### *General Section*

The **General** section allows you to enter general information about the node, such as:

- **Label** - Unique name by which an element will be referred to in reports, error messages, and tables.
- **X (Easting)** - The location of the node may be represented as an X-value or an Easting value, depending on individual preferences.

- **Y (Northing) -** The location of the node may be represented as a Y-value or a Northing value, depending on individual preferences.
- **Ground Elevation -** Elevation of the ground surface at the node.
- **Station / Calculated Station** - Distance along the alignment of pipes. The starting station can be specified at an outlet or wet well, and is calculated for nodes and junctions.

> **The Station attribute is only editable for Outlet and Wet Well elements. For all other elements, the station is computed during a calculation based on the initial station and pipe lengths.**

### *Wet Well Hydraulic Summary Section*

This section reports the hydraulic grade line exiting the wet well, and hence the level in the wet well itself. The summary consists of **Hydraulic Grade Line Out**, which represents the hydraulic grade in the wet well calculated as explained in the *Hydraulic Transition from Gravity to Pressure Network* section in Appendix B.

### *Flow Summary Section*

This section reports a summary of the total flow at a node and the distribution of this flow.
For **Steady State** mode the fields displayed are as follows:

- **Total Dry Weather Flow** - Total flow at a node resulting from sewage generated during dry weather from the network upstream of a given node.
- **Total Wet Weather Flow** - Total flow at a node resulting from the intrusion of rainfall water into the sewer system from the upstream network. Wet weather load consists of groundwater infiltration and rainfall inflow. Groundwater infiltration occurs in gravity pipes while inflows occur at manholes and wet wells.
- **Pumped Flow** - Portion of the total flow that comes from pumps located in the upstream network, when the **Use Pumped Load** toggle in the **Calculation Options** dialog is On. In this case, the flow coming out of pumps is treated as a fixed flow, to which peaking factors do not apply.
- **System Known Flow** - Portion of the total flow derived from manually entered Known Flows upstream of the reporting point. Known flows are not additive except at network junctions. The Known Flow component will remain constant until it encounters a downstream Known Flow of a different value.
- **Total Flow** - Total flow going through an element of the network.

For **EPS** mode the fields displayed are as follows::

- **Total Flow -** Represents the total flow coming to the structure, which includes the upstream flow through pipes, incoming diverted flow, and the local sanitary loads and inflows.
- **Diverted Flow** - Represents the amount of the total flow, which is diverted from the inlet to the diversion target.
- **Flow In -** Represents the portion of the flow established locally at the selected node, such as unit dry weather and wet weather loads.

> **You can toggle between Steady State and EPS modes by clicking the GO button and changing the Calculation Type.**

## Pumps

### *Pump General Tab*

The **General** tab for pumps is organized into the following groups:

- **General** - General data about the pump.
- **Pump** - Contains the type of pump curve and related data.
- **Initial Setting** - Initial conditions describing the pump's behavior at the start of the analysis.
- **Pipes** - Indicates the direction in which the pump is operating (from upstream node to downstream node), and lets you reverse the direction of pumping by clicking the **Reverse** button.
- **Calculated Hydraulics** - Reports the hydraulic grade and pressure at the adjacent end of both connecting pipes, intake and discharge.
- **Operating Point** - Represents the values for pump head and discharge, which are computed by the program to balance with the remaining system heads and flow rates.

**The Initial Settings are used as the permanent settings. However, they can be overruled by the presence of controls if the Use Controls in Steady State Analysis check box in the Calculation Options dialog is checked.**

For further details, refer to the topics describing each section.

### *General Section*

This section allows you to enter general information about the pump such as:

- **Label** - Unique name referencing the pump in reports, error messages, and tables.
- **X (Easting)** - The location of the pump may be represented by an X-value or an Easting value, depending on individual preferences.
- **Y (Northing)** - The location of the pump may be represented by a Y-value or a Northing value, depending on individual preferences.
- **Elevation** - Elevation of the pump.

### *Pump Section*

The information required for a pump depends on the type of pump that is selected. The possible information is as follows:

- **Pump Type -** Select one of the six available types of pump curves.
- **Pump Power -** Represents the water horsepower, or horsepower that is actually transferred from the pump to the water. Depending on the pump's efficiency, the actual power consumed (brake horsepower) may vary.
- **Shutoff -** Point at which the pump will have zero discharge. It is typically the maximum head point on a pump curve.
- **Design -** Point at which the pump was originally intended to operate. It is typically the best efficiency point (BEP) of the pump. At discharges above or below this point, the pump is not operating under optimum conditions.
- **Max Operating -** Highest discharge for which the pump is actually intended to run. At discharges above this point, the pump may behave unpredictably, or its performance may decline rapidly.

- **Max Extended -** Absolute maximum discharge at which the pump can operate, adding zero head to the system. This value may be computed by the program, or entered as a custom extended point.

**All defined pump curve points have an associated head and discharge.**

### *Initial Setting Section*

The initial conditions for a pump describe the pump's behavior at the start of the analysis. These conditions include:

- **Status** - One of two available status conditions: On (normal operation), Off (no flow under any condition).
- **Relative Speed Factor** - Characteristics of the pump relative to the speed for which the pump curve was entered, in accordance with the affinity laws. A speed factor of 1.00 will indicate pump characteristics identical to those of the original pump curve.

**In Steady-State Analysis mode, the Pump Status is used as the permanent status. However, it can be overruled by the presence of controls, if the Use Controls in Steady-State Analysis check box in the Calculation Options dialog is checked. The Calculation Options dialog is accessed by clicking the GO button in the main view to display the Calculation tab of the Scenario Editor, and then clicking the Options button.**

### *Pipes Section*

This indicates the direction in which the pump is operating (from upstream pipe to downstream pipe).

**You can switch the Upstream and Downstream Pipes by clicking the Reverse button.**

### *Calculated Hydraulics Section*

- **Intake Pump Grade -** HGL on the suction side of the pump.
- **Intake Pump Pressure** - Pressure on the suction side of the pump.
- **Discharge Pump Grade** - HGL on the downstream side of the pump.
- **Discharge Pump Pressure** - Pressure on the downstream side of the pump.

### *Operating Point Section*

The pump's operating point represents the values for pump head and flow, which are necessary to meet the discharge load on the pump as well as overcome the system losses.

The calculated parameters are:

- **Pump Head** - Head generated by the pump at the operating point.
- **Pressure Flow** - Flow through the pump.

**For a constant power pump, the calculated operating point may be outside the normal operating range of a realistic pump. Be very cautious and check all results carefully.**

**For more information about the theory behind the pump operating point, see the help on pump theory in Appendix B.**

## Pressure Junction

### *Pressure Junction General Tab*

The **General** tab for pressure junctions is organized into the following sections:

- **General** - General information about the junction.
- **Hydraulic Results -** Calculated hydraulic grade and pressure at the junction.

For further details, refer to the topics describing each section.

### *General Section*

The **General** section for pressure junctions allows you to enter general information such as:

- **Label** - Unique name by which a pressure junction will be referred to in reports, error messages, and tables.
- **X (Easting)** - May be presented as an X-value or as an Easting value, depending on individual preferences.
- **Y (Northing)** - May be presented as a Y-value or a Northing value, depending on individual preferences.
- **Elevation** - Elevation of the pressure junction.

### *Hydraulic Results Section*

The **Hydraulic Results** section for pressure junctions reports the following results:

- **Calculated Hydraulic Grade Pump** Hydraulic grade at the junction.
- **Pressure** - The pressure calculated at this junction.

## Gravity Pipe

### *Gravity Pipe General Tab*

The **General** tab for gravity pipes contains all the physical data necessary for successfully modeling pipes. The tab also provides a brief hydraulic summary of the flow and velocity through the pipe, as well as the pipe’s constructed slope and full flow capacity. This tab has five sections:

- **Pipe** - General characteristics of the pipe.
- **Invert Elevations** - Upstream and downstream invert elevations.
- **User Defined Length** - Specify whether the pipe length is calculated automatically or user-defined.
- **User Defined Bend Angle** - Specify whether the angle the pipe creates with the downstream pipe is calculated automatically or user-defined.
- **Hydraulic Summary** - Displays the computed hydraulic data of the pipe.

For further details, refer to the topics describing each section.

### *Pipe Section*

This section is where all of the pipe general characteristics are entered. The following fields are available:

- **Label** - Unique name by which a pipe will be referred to in reports, error messages, and tables.
- **Section Shape** - Select one of the following pipe section shapes: arch, box, circular, or elliptical (vertical or horizontal).

- **Material** - Pipe material with its associated roughness value selected from those available in the **Material Library**.
- **Roughness Coefficient** - Specify the coefficient corresponding to the roughness method selected during the project setup (Manning's n, Kutter's n, Hazen-Williams C, or Darcy-Weisbach roughness height) for the selected material. You can keep the roughness value associated with the selected material in the **Material Library**, or override the roughness value for that specific pipe.
- **Section Size** - Display a section size from the list defined in the **Section Size Library**.
- **Number of Sections** - Number of identical, parallel pipe sections used in the hydraulic calculations.

> **By clicking the ellipsis (...) button located next to the Material field or the Section Size field, you can access the respective engineering library to create and customize materials and section sizes.**
>
> **You can let the program choose a section size for you during an automatic design calculation.**

### *Invert Elevations Section*

In this section, you select whether upstream and downstream pipe inverts will be entered by you or set to the sump elevation of the upstream and downstream node. If **Set Invert to Upstream/Downstream Structure** box is not checked, you can set the upstream and downstream invert elevations in this section. Otherwise, the program will compute the pipe upstream/downstream invert elevations using the upstream/downstream structure sump elevation and desired sump depth.

### *User Defined Length Section*

If the **User Defined Length** box is checked, you can enter a pipe length. Otherwise, the program will compute a pipe length based on the drawn alignment of the pipe. User-defined lengths are useful for drawing quick schematics to speed along your design process.

### *User Defined Bend Angle Section*

If the **User Defined Bend Angle** box is checked, you can enter an appropriate bend angle for the purpose of calculating loss through a junction or inlet structure. If the box remains unchecked, the program will calculate the bend angle across the junction or inlet structure based on the alignment of the pipe relative to the downstream pipe.

### *Hydraulic Summary Section*

Here you can view the calculated hydraulic characteristics of a pipe, which include:

- **Average Velocity** - Average velocity of the flow in the pipe, calculated by using one of the available average velocity methods.
- **Constructed Slope** - The difference in the invert elevations between the upstream and downstream end of the pipe divided by its length.
- **Full Capacity** - Computed discharge in the pipe when it is flowing full.
- **Design Capacity** (Steady State only) - Computed discharge in the pipe, based on the Design Percent Full value specified in the **Design** tab.
- **Excess Full Capacity** (Steady State only) - Difference between the full flow capacity of the pipe and the actual calculated flow in the pipe.
- **Excess Design Capacity** (Steady State only) - Difference between the design capacity of the pipe and the actual calculated flow in the pipe.
- **Total Flow** - Total flow in the pipe during the run.

## Pressure Pipe

### Pressure Pipe General Tab

The **General** tab for pressure pipes is organized into the following groups:

- **Pipe** - General pipe data.
- **Initial Status** - Specify whether the pipe is initially open or closed.
- **Invert Elevations** - Displays elevations of the pipe inverts at the upstream (*From node*) and downstream (*To node*) end.
- **User-Defined Length** - Specify whether the pipe length is calculated automatically or user-defined.
- **Nodes** - Displays the nodes at the upstream and downstream ends of the pipe.
- **Hydraulic Results** - Displays calculated hydraulic data.

For further details, refer to the topics describing each section.

### Pipe Section

In this section you enter in all of the pipe general characteristics:

- **Label** - Unique name referencing the pipe in reports, error messages, and tables.
- **Material** - Pipe material, with its associated roughness value, selected from the **Material Library**.
- **Diameter** - Diameter of the pipe.
- **Roughness Coefficient** - Pipe roughness coefficient or value associated with the roughness method selected during the project setup (Manning's n, Hazen-Williams C, or Darcy-Weisbach roughness height) for the selected material. You can keep the roughness value associated with the selected material, as defined in the material library, or override the roughness value for that specific pipe.
- **Minor Loss Coefficient** - Coefficient K used in the minor loss equation, as defined in the *Minor Losses* section in Appendix B. This is the equation most commonly used for determining the headloss in a fitting, valve, meter, or other localized component.
- **Check Valve** - When this box is checked, flow can only travel from the *From Node* to the *To Node* in a pressure pipe.

**By clicking the ellipsis (...) button located next to the Material you can access the engineering library to create and customize materials.**

**By clicking the ellipses (…) button on the Minor Loss Coefficient field, you can access the Minor Loss elements and generate composite minor loss coefficients to be applied to the pressure pipe.**

**Set the minor loss coefficient value to 0.0 if there is no minor loss in the pipe.**

### Minor Loss Elements

Pressure pipes can have an unlimited number of minor loss elements associated with them. This program provides an easy-to-use table for editing these minor losses. The minor loss table consists of four columns:

- **Quantity** - The number of minor losses of the same type to be added to the composite minor loss for the pipe.
- **Minor Loss** - The type of minor loss element.
- **K Each** - The headloss coefficient for a single minor loss element of the specified type.

- **K Total** - The total minor loss coefficient for the row. It is the **Quantity** multiplied by the **K Each**.

The **Minor Loss Elements** dialog also has three command buttons:

- **Insert** - Insert a row in the table.
- **Duplicate** - Create a new row in the table with the same values as the selected row.
- **Delete** - Delete the selected row of the table.

The **Minor Loss Elements** dialog is accessed by clicking the **ellipsis (…)** button next to the **Minor Loss Coefficient** choice list on the **Pressure Pipe Editor**.

### *Initial Status Section*

The initial status of the pipe can be either Open or Closed. The status can possibly change when calculations are performed based on the presence of controls for that pipe.

**In Steady State Analysis mode, the Initial Status is used as the permanent status. However, it can be overruled by the presence of controls, if the Use Controls in Steady-State Analysis check box in the Calculation Options dialog is checked. The Calculation Options dialog is accessed by clicking the GO button in the main view to display the Calculation tab of the Scenario Editor, and then clicking the Options button.**

### *Invert Elevations Section*

The **From Node** and **To Node Elevations** of the pressure pipe can be entered and viewed here. . If **Set Invert to Upstream/Downstream Structure** box is not checked, you can set the **From** and **To** elevations in this section. Otherwise, the program will compute the pipe's **From** and **To** elevations using the upstream/downstream gravity structure's sump elevation.

**Disabled elevations are defined at the connecting node structure. Only inverts that are connecting to gravity node structures such as wet wells or outlets can be edited. Inverts connecting to pressure elements such as pressure junctions and pumps cannot be edited.**

### *User-Defined Length Section*

If the **User-Defined Length** box is checked, you can enter a pipe length. Otherwise, the program will compute a pipe length from node center to node center, accounting for bends if there are any. Creating user-defined lengths is useful for drawing quick schematics to accelerate your design process.

### *Nodes Section*

Displays the nodes that are upstream (*From node*) and downstream (*To node*) of the pipe. If the flow is traveling from the **From Node** to the **To Node** then the flow value will positive. If flow is traveling from the **To Node** to the **From Node** it will be negative. You can reverse the **From Node** and the **To Node** by clicking the **Reverse** button.

### *Hydraulic Results Section*

This section reports the following hydraulic results:

- **Pressure Flow** - Calculated total flow in the pipe.
- **Velocity** - Calculated velocity in the pipe.
- **Headloss Gradient** - Headloss in the pipe represented as a slope, or gradient.
- **Pressure Pipe Headloss** - Loss of energy in the pipe due to friction and minor losses.

- **Control Status** - Open or closed status of the pipe. Open means that flow occurs in the pipe and closed means that there is no flow.

## 6.3.2 Headlosses Tab

The **Headloss** tab is used to specify the method and parameters that are used to calculate the headlosses through any structure located at a junction in a gravity network. The Standard, AASHTO, Generic, or HEC-22 Energy methods are available to automatically calculate headloss based on structure geometry and flows, or a desired headloss value can be specified directly using the Absolute method. Furthermore, the model results for the exit pipe, which are often of interest in computing headlosses, can be seen here under the Exit Pipe Summary. This tab contains three sections:

- **Headlosses** - The headloss method can be selected from the list box from the following choices:
  - Absolute
  - Standard
  - HEC-22 Energy
  - AASHTO
  - Generic
- **Headloss Method Parameters** - Parameters dependent upon which headloss method is chosen.
- **Exit Pipe Summary** - Calculated hydraulic properties of the pipe downstream of the junction.

### Headlosses Section

The various headloss methods can be selected from the choice list. The methods include the Absolute, Standard, Generic, AASHTO, and HEC-22 methods. Refer to Appendix B for more information regarding the theory underlying these methods.

### Headlosses Method Section

The items displayed are dependent upon which method is chosen in the **Headlosses** section. The choices are as follows:

- **Absolute** - Enter the desired value for headloss at the structure. This method ensures that the headloss across the structure will be equal to this value regardless of the actual flows or geometry of the structure.
- **Standard** - Enter the headloss coefficient for the structure. The headloss across the structure will be equal to this value multiplied by the exit pipe velocity head.
- **HEC-22** - The HEC-22 Energy method assumes the energy loss across a structure is proportional to the velocity head of the exit pipe. The proportionality constant takes into consideration several variables based on structure shape, configuration, plunging flows, and benching of the structure bottom. An in-depth description of the theory can be found in the *Junction Headlosses* section Appendix B of the help, and in Chapter 7 of the *HEC-22 Urban Drainage Design Manual.* All but one aspect of the proportionality constant, benching, is determined internally based on known variables such as structure type, pipe sizes, and angles between pipes. The benching method is user-defined, and is selected from the list box in the **HEC-22 Benching Method** field. The following four benching methods are available:
  - **Depressed** - The floor elevation of the structure is lower than the invert elevation of the exit pipe.
  - **Flat** - The floor elevation of the structure is equal to the invert elevation of the exit pipe.

- **Half** - The floor elevation of the structure is equal to the elevation at the center of the exit pipe.
- **Full** - The floor elevation of the structure is equal to the elevation at the top of the exit pipe.

- **AASHTO** - The AASHTO headloss method considers several variables when computing the headloss across a structure. All of these variables can be determined internally except for the shaping of the structure. You select the shaping method (None or Full) from the choice list. See the *Junction Headlosses* section of Appendix B of the help for a discussion of the theory underlying the AASHTO method.
- **Generic** - The input values used by this method to compute the headloss across a gravity structure are the upstream and downstream coefficients. The loss across the structure is computed as the downstream coefficient multiplied by the velocity head in the downstream pipe minus the upstream coefficient times the velocity head in the governing upstream pipe. Note that if a value of zero is specified as the value for the upstream coefficient this method will produce the same results as the Standard Method. Refer to Appendix B for more information regarding this method.

#### Exit Pipe Summary Section

This area shows the following calculated properties of the exit pipe, which are typically of interest in determining structure headloss:

- **Exit Discharge** - Discharge in the pipe downstream of the structure.
- **Exit Velocity** - Velocity at the upstream end of the pipe downstream of the structure.
- **Exit Velocity Head** - Velocity head at the upstream end of the pipe downstream of the structure.

### 6.3.3 Diversion Tab

The **Diversion** tab is used to define the characteristics of the diversion structure. At this tab you select whether this element will be a diversion or not. If the **Has Diversion** box is checked on, flows going out of this element will be diverted according to the parameters defined on this tab. This tab contains five sections:

- **Has Diversion** - Check box indicating whether this element should divert flows or not.
- **Diversion Parameters** - Diversion type (overflow or a diversion) and Diversion Target are defined here.
- **Diversion Rating Curve Table** - Table defining diverted flows as a function of upstream (system) flows.
- **Flow Diverted Out** - Summary of the flows diverted out of the element at the diversion.
- **Flow Diverted In** - Summary of the flows diverted to this element from other diversions in the system.

#### Diversion Parameters

The user can specify if and where diverted flow will reenter the model in this section of the **Diversion** tab. The user can choose between an overflow diversion target <Overflow> and a diversion target element. The program will compile a list of available diversion targets that will not create a loop in the system. An element is available as a diversion target if it is downstream of the diversion element. Any element in another network can be used as a diversion target, provided that the target network is downstream from the origin network. A target network is considered to be downstream of the origin network if there is no way for water to get from the target network to the origin network.

You will not be able to edit the **Diversion Target field** until the box labeled **Has Diversion** is checked.

> **If diversions create a loop within the system a validation message will pop up at the calculation time.**

### Flow Diverted Out

This section of the **Diversion** tab provides the user with a summary of results for the flows diverted out of the element.

- **Percent Diverted Out** - Percentage of total flow coming to the diversion element that is diverted out of this diversion element.
- **Diverted Flow Out** - Total flow that is diverted out of this diversion element.
- **Non-Diverted Flow Out** - Total flow that is not diverted out of this diversion element. This is the flow that goes to the downstream pipe.
- **Total Flow Out** - The total flow leaving the structure. This includes both the flow that is diverted and the flow leaving through the downstream pipe.

> **The sum of the Diverted Flow Out and Non-Diverted Flow Out is equal to the Total Flow Out.**

### Flow Diverted In

This section of the **Diversion** tab provides the user with a summary of results for the flows diverted to this element from other diversions in the system.

- **Local Diverted Flow In** - Sum of the flows diverted to this element from upstream elements in the same network.
- **Global Diverted Flow In** - Sum of the flows diverted to this element from diversions in other networks.
- **Total Diverted Flow In** - Total flow that is diverted to this element.

> **Flow will be diverted to an element if the element is selected as a Diversion Target. Unlimited number of diversions can divert flow to a single element.**

### Diversion Rating Curve Table

This section of the **Diversion** tab is used to specify the diversion rating curve. A diversion rating curve defines diverted flows as a function of the total system flow. You must specify at least two points in the **Diversion Rating Curve Table**. For each point, diverted flow must be smaller than the system flow. The program uses linear interpolation/extrapolation to determine the values of diverted flows that lie between the points in the table or outside of the range of the table. If the computed value of the diverted flow is negative, the program will set the diverted flow to 0.

To be able to edit the **Diversion Rating Curve Table** you must check the box labeled **Has Diversion**.

## 6.3.4 Controls Tab

### Pump and Pressure Pipe Controls Tab

Controls allow you to configure the hydraulic model by changing the pump or pipe settings when specific junction pressures or wet well water levels occur in the network. The following buttons are available:

- **Add** - This will open the **Control** dialog, where a new control can be added and the specifics can be entered.
- **Edit** - Select the description of the control you wish to edit, and click this button. This will open the **Control** dialog, where the specifics can be edited.
- **Duplicate** - Duplicates an existing control. Select the description of the control you wish to duplicate, and click this button.
- **Delete** - Deletes an existing control. Select the description of the control you wish to delete, and click this button.

**Pipes with check valves cannot have controls.**

## Control Dialog

Several types of information are required to define a control for a pressure pipe or pump. This data is grouped into the following sections:

- **Preview** - Textual description of the control being edited.
- **Control** - Specify the type of control, either status or setting.
- **Control Condition** - Specify the controlling node and the control setting.

### *Control Preview Section*

The **Preview** section provides a textual description of the control being edited. The control preview is continuously updated while you edit a control.

### *Control Section*

This software supports two types of controls:

- **Status** - Controls the Open/Closed status for pipes, or the On/Off status for pumps.
- **Setting** - Controls the relative speed factor of a pump.

**Only status controls are available for pipes. Setting controls are not appropriate.**

**When pumps are turned on by a control, their relative speed factor is set to 1.00.**

**To activate a closed or inactive valve, use a setting control. Similarly, to turn a pump on at a relative speed setting other than 1.00, use a setting control.**

### *Control Condition Section*

A control can be triggered by a specified pressure or hydraulic grade being reached in any wet well or pressure junction, or they can also be triggered based on the time during an analysis. This section contains the following fields:

- **Condition** - You can specify a time based or node based condition.

When **Node** is the selected condition, the control will be triggered based on the hydraulic grade or pressure at the selected pressure junction or wet well, and you must specify the following information.

- **Node** - Specifies the controlling node.

- **Comparison** - Triggers the control when the specified junction or wet well's hydraulic parameter is above or below the node condition's hydraulic parameter.
- **Hydraulic Grade or Pressure** - The control conditions at the control node can be expressed in terms of hydraulic grade or pressure. Depending on which is chosen, the software will display the calculated value of the other.

**Example:** Closed when node J-2 below 10 psi - means that when the pressure at junction J-2 is below 10 psi, the controlled pipe will close.

When **Time** is the selected condition, the control will be triggered when the selected time is reached during an EPS. You must specify the following information:

- **Time** - When this time is reached during an EPS the control will be triggered.

**Example:** Close at time 4.00 hr - means that when the simulation reaches hour 4 the controlled pipe will close.

### 6.3.5 Profile Tab

This tab contains information about the physical properties and calculated profile of the flow through the pipe. This tab is divided into three sections:

- **Upstream Elevations** - Displays elevations related to the upstream end of the pipe.
- **Downstream Elevations** - Displays elevations related to the downstream end of the pipe.
- **Profile** - Displays the type of profile exhibited by the flow through the pipe, and the energy slope and headloss in the pipe.

#### Pipe Elevations Section

Elevation information is grouped into two sections on the **Profile** tab, upstream on the left and downstream on the right, related to the two pipe ends. The upstream and downstream elevation information includes:

- **Ground elevation** - Elevation of the ground surface at the node, edited in the **Element Editor** for that node.
- **Cover** - Distance between the crown (soffit) of the pipe and the ground surface elevation.
- **Crown** - Elevation of the top of the pipe section.
- **Hydraulic Grade** - Hydraulic grade at the ends of the pipe.
- **Depth** - Flow depth at the ends of the pipe.
- **Invert** - Elevation of the pipe invert.

#### Pipe Profile Section

This section contains the following information:

- **Description** - Displays the profile type that the flow exhibits as it travels through the pipe. In the case of a composite profile, this section will display all the profile types.
- **Energy Slope** - The result of dividing the headloss per pipe length.
- **Headloss** - The total headloss through the pipe.
- **d/D (depth/Rise) -** A ratio of the average depth ((Depth Upstream + Depth Downstream) / 2) in the pipe to the pipe’s rise.

## 6.3.6 Design Tab

### Manholes, Junction Chambers, and Outlets

#### *Design Tab (for Manholes, Junction Chambers, and Outlets)*

The **Design** tab provides an interface for entering the nodal constraints for SewerCAD's automated design feature. Manholes, junction chambers, and outlets all support the following groups of constraints:

- **Local Pipe Matching Constraints** - If you toggle this on, you can enter pipe matching constraints specific to this junction structure.
- **Design Structure Elevation** - Specifies if the program is to design the structure's sump elevation during an automatic design.

#### *Local Pipe Matching Constraints Section*

Check the box in this section if you want to specify pipe matching constraints for the structure that are different from the Design Alternative's default values. During an automatic design, the program will adjust the elevations of the pipes adjacent to the structure according to the structure's matching constraints. The two choices for matching are Inverts and Crowns. Additionally, the downstream pipe can be offset from the upstream pipe(s) by a specified amount. This value is called the Matchline Offset. The program also supports the design of drop structures. In some situations, drop structures can minimize pipe cover depths while maintaining adequate hydraulic performance. This can be done by clicking the **Allow Drop Structure** check box.

#### *Design Structure Elevation Section*

Check the box in this section if you want the structure's sump elevation adjusted during an automatic design. If this box is checked, the **Desired Sump Depth** field becomes editable. The sump depth is the distance below the lowest pipe invert.

### Gravity Pipe

#### *Pipe Design Tab*

The **Design** tab for pipes allows you to customize the automatic design process for a particular pipe. You can specify whether the pipe will be designed, and, if so, how the automated design process should be constrained. This tab is divided into the following sections:

- **Design Pipe** - If this box is checked, the program will automate the design of the pipe.
- **Part Full Design** - Allows you to tell the program to design the pipes in the system so the depth of flow is a percentage of the pipe diameter.
- **Allow Multiple Sections** - Allows the automatic design process to use several identical pipes in parallel.
- **Limit Section Size** - Allows you to limit the section sizes from the **Section Size Library** to be used during automatic design process.
- **Range Constraints** - Displays the desired cover, slope, and velocity conditions for the pipe design process.

#### *Design Pipe Section*

If the **Design Pipe** box is checked, the program will automate the design of the pipe. You can also select whether the program should adjust the upstream and downstream invert during the design process by checking the **Design Upstream Invert** and **Design Downstream Invert** boxes, respectively. If these boxes are not checked, the inverts will reflect the user-defined values entered under the **Pipe General** tab. Finally, you can choose to specify local design constraints by checking the box labeled **Specify Local**

**Constraints**. This will enable the fields in the **Range Constraints Section** and allow you to set constraints specific to the pipe. These local constraints override the default design constraints set through the **Analysis** menu.

#### *Part Full Design Section*

If the **Specify Local Constraints** box in the **Design Pipe** section is checked, you can specify the **Design Percent Full** target to be used by the design algorithm. Thus, pipes may be sized such that the depth of flow is a percentage of the pipe diameter.

#### *Allow Multiple Sections Section*

If the **Specify Local Contraints** box in the **Design Pipe** section is checked, you can choose to let the design algorithm adjust the number of sections in parallel, up to the specified **Maximum Number Sections**.

#### *Limit Section Size Section*

If the **Specify Local Contraints** box in the **Design Pipe** section is checked, you can limit the pipe section height to a **Maximum Section Rise** value during the design process.

#### *Range Constraints Section*

If the **Specify Local Constraints** box in the **Design Pipe** section is checked, you have the option to set the boundary conditions for the pipe design process. You can set the following design constraints: **Minimum** and **Maximum Velocity**, **Minimum** and **Maximum Cover**, and **Minimum** and **Maximum Slope**.

## 6.3.7 Section Tab

### Section Tab (for Wet Wells)

The wet well section data includes the information necessary to describe the storage characteristics of a wet well. These characteristics have been factored into the following groups:

- **Section** - The type of cross section and the basic storage parameters.
- **Cross Section** - Parameters describing the cross sectional geometry.
- **Operating Range** - The minimum, initial, and maximum operating elevations.

### Wet Well Section Section

The general information under the **Section** heading for wet wells is:

- **Section** - Choose the type of cross section for the wet well. There are two types of cross sections to choose from - Constant Area and Variable Area.
- **Inactive Volume** - Enter the inactive volume for this wet well, as defined in the illustration on this dialog.
- **Total Active Volume** - If this is a Constant Area wet well, the total active volume will be computed from the other wet well data and this field will not be editable. If this is a Variable Area wet well, enter the total storage volume for the wet well.

**Wet well section characteristics do not affect the results or simulation during a Steady-State Analysis. The volume and storage capacity of the wet well are utilized during an Extended Period Simulation.**

### Cross-Section Section

The two basic types of wet wells are Constant Area and Variable Area. The **Cross Section** section changes depending on which type is chosen. The alternatives are as follows:

- **Constant Area** - The cross sectional geometry of the wet well is constant between the minimum and the maximum operating elevations. Two parameters are needed to fully describe a constant area wet well section:
  - **Cross Section** - Choose whether the cross section is circular or non-circular.
  - **Average Area/Diameter** - Enter the average area of the non-circular cross section, or the diameter of the circular cross section.
- **Variable Area** - The cross sectional geometry of the wet well varies between the minimum and maximum operating elevations. The following input is available to describe the variable area:
  - **Depth Ratio/Volume Ratio Table** - Enter a series of points describing the storage characteristics of the wet well. For example, at 0.1 the total depth (depth ratio = 0.1) the wet well stores 0.028 the total active volume (volume ratio = 0.028).

**The storage characteristics of the wet well can be plotted. Choose Tank Curve from the Report Button at the bottom of the Wet Well dialog.**

**Wet well section characteristics do not affect the results or simulation during a Steady-State Analysis. The volume and storage capacity of the wet well are utilized during an Extended Period Simulation.**

### Operating Range Section

This is where you can set the absolute limits for the water levels in a wet well. The range can be defined in terms of Elevations or Levels. Elevations are relative to the same datum as the rest of your system, while levels refer to heights of water above the wet well's base elevation. The associated fields prompt you for the following values:

- **Maximum** - This is the highest water surface elevation or level in the wet well.
- **Alarm** - If the HGL in the wet well goes above the alarm elevation or level during the analysis then a warning message will be generated during that time step.
- **Initial** - The use of this parameter depends on whether the **Fixed Level** toggle is set to On or Off during a Steady-State Analysis. Refer to the *Hydraulic Transition from Gravity to Pressure* topic in Appendix B for a further explanation. During an Extended Period Simulation this value represents the starting elevation at the beginning of the simulation.
- **Minimum** - Lowest water surface elevation or level in the wet well.
- **Base** - The elevation of the base of the wet well.
- **Fixed Level** - When this checkbox is checked, the level in the tank will not be adjusted to reflect the tank inflow during a **Steady State** analysis, and the initial level described above will be used as the HGL. The **Fixed Level** checkbox has no effect when running an EPS analysis.

**The initial elevation must be between the maximum and minimum elevations.**

## 6.3.8 Loading Tab

SewerCAD classifies loads as sanitary (dry weather) loads, wet weather loads, and known flows.

Sanitary loads correspond to loads produced by residential, commercial, recreational, and industrial activity. A sanitary load represents the base load to the sewer system. Wet weather loads are related to rainfall activity. They are caused by groundwater infiltration and rainfall inflow.

The **Loading** tab for manholes, wet wells, and pressure chambers is organized into the following groups:

- **Sanitary (Dry-Weather) Flow** - Table containing a collection of sanitary loads.
- **Inflow -** Table containing a collection of wet weather loads, and is accessible only for wet wells and manholes.
- **Known Flow -** Used to model a local load entering a manhole or wet well if, for instance, you have computed the hydrology using an external method such as the TR-55 tabular method. The **Known Flow** field is accessible only for wet wells and manholes.

### Sanitary (Dry-Weather) Flow Section

The **Sanitary Flow** section is specified as a collection of sanitary loads applied to the selected node. The following types of loads can be applied:

- **Hydrograph - Flow vs. Time** - A flow vs. time distribution
- **Unit Load - Unit Type & Count -** The type of **Unit Load** and the number of units associated with **Unit Load**. For example, 5000 passengers at an Airport.
- **Pattern Load - Base Flow & Pattern -** A direct, known sanitary load with a set pattern.

The following operations can be performed on the **Sanitary Flow** list in the section:

- **Add** - Add a new load to the **Sanitary Flow** list.
- **Edit** - Opens the editor for an existing load in the list.
- **Delete -** Deletes a sanitary load from the list.
- **Pie Chart** - Generates a pie chart that depicts the distribution of the sanitary loads in the list.

### Inflow Section

The **Inflow** section is specified as a collection of wet weather loads applied to the selected node. The following types of loads can be applied.

- **Hydrograph - Flow vs. Time** - A flow vs. time distribution
- **Pattern Load - Base Flow & Pattern -** A base wet-weather load with a set pattern.

The following operations can be performed on the **Inflow** list in the section:

- **Add** - Add a new load to the **Inflow** list.
- **Edit** - Opens the editor for an existing load in the list.
- **Delete -** Deletes an inflow load from the list.

**The Inflow section is only available for wet wells, pressure junctions, and manholes.**

### 6.3.9 Infiltration Tab

This tab allows you to enter and view the infiltration data (direct and additional) for a gravity pipe. The following sections are available:

- **Infiltration** - Infiltration can be estimated proportionally to gravity pipe characteristics such as length or number of pipe defects. Depending on the **Infiltration Type** specified, an additional section appears containing fields for entering the infiltration loading unit and rate, pattern load, or hydrograph information.
- **Local** - Contains a field for displaying the local infiltration as specified in the previous sections, as well as an **Additional Infiltration** field for entering a lump sum infiltration.
- **System** - Displays the cumulative infiltration resulting from the upstream pipe network.

#### Infiltration Section

Four types of direct infiltration are supported, as specified in the **Infiltration Type** choice list:

- Pipe Length
- Pipe Diameter-Length
- Pipe Surface Area
- Count Based (this may be used to account for infiltration proportional to the number of pipe defects)

Direct infiltration is defined by:

  - **Infiltration loading unit** - Unit that is used to define infiltration rate.
  - **Infiltration rate per loading unit** - Rate of infiltration for each infiltration loading unit.

In addition to the four direct infiltration types, the following time-based loading types can be selected from the **Infiltration Type** list as well:

- Hydrograph
- Pattern Load

#### Infiltration Hydrograph

SewerCAD also allows infiltration to be defined as a direct hydrograph into the gravity pipe. When **Hydrograph** is chosen from the **Infiltration Type** pull-down menu, a table will appear where time/flow data can be added.

The following operations can be performed on the table:

- **Insert** - Inserts a row in the table above the selected row
- **Duplicate** - Duplicates the selected row
- **Delete** - Removes the selected row from the table

#### Infiltration Pattern Load

A Pattern Load is defined by a single base load and a pattern, which is a series of multipliers that define how the base load is distributed over time.

If the **Pattern** is set to fixed then the infiltration load is constant over time. You can enter the **Pattern Manager** by clicking the **ellipses (…)** button in the **Pattern** field.

When running a Steady-State analysis the infiltration load is equivalent to the base load regardless of the pattern.

### Local Section

This section consists of the following fields:

- **Local Infiltration** - The infiltration entering the pipe as defined in the **Infiltration** section.
- **Additional Infiltration** - Used to specify a lump sum infiltration amount.
- **Total Infiltration** - The sum of the additional infiltration and direct infiltration.

### System Section

This section consists of the following fields:

- **System Infiltration** - The cumulative infiltration resulting from the upstream pipe network.
- **System Additional Infiltration** - The cumulative additional infiltration resulting from the upstream pipe network.
- **System Total Infiltration** - The cumulative total infiltration resulting from the upstream pipe network.

## 6.3.10 Cost Tab

On this tab, you can specify whether or not the element is to be included in the cost analysis. If the element is selected to appear in the cost analysis then you can enter the costs associated with the element. This tab is comprised of the following components:

- **Include in Cost Calculation?** - A check box that allows you to control whether or not this element will be included in the cost analysis. If this box is checked, the element will be included in the cost calculation.
- **Construction Costs** - Contains a table for an element for entering cost items that can be expressed in terms of a quantity, unit, and unit cost.
- **Non-Constructions Costs** - Contains a table for entering costs related to the elements that need to be expressed as either a lump sum or as a percentage of the construction costs.

### Include In Cost Calculation?

This check box allows you to control whether or not this element will be included in the cost calculation. If this box is checked, the element will be included in the cost calculation. If you are modeling a new subdivision, most of the elements in your model will probably be included in the cost calculation. However, if you are adding onto an existing system, you may only calculate the cost for a small portion of the total elements in your system.

The value of this field can be varied by alternative. This can be useful if you want to compute the costs for different portions of your system separately. For instance, if you have several phases of construction that you want to cost separately, you could create one cost alterative that only includes elements in phase one, and another alternative that only includes elements in phase two. When you perform your cost analysis, you can then get cost reports detailing each phase of construction.

### Non-Construction Costs

The **Non-Construction Costs** section of the **Cost** tab contains a table that allows you to enter an unlimited number of non-construction cost items for each element. A non-construction cost item can be specified as either a lump sum value or as a percentage of the total construction costs for the element. Each non-construction cost contains the following four components.

  - **Label** - A unique name that identifies the non-construction cost item. The labels must be different for all non-construction cost items in a table.

- **Factor** - A numeric value that is used in conjunction with the operation to compute the cost for a non-construction cost item.
- **Operation** - The operation that will be applied against the factor to compute the total cost for the non-construction cost item. The two possible values for this field are lump sum or percentage of the total construction costs.
- **Cost** - The cost of the non-construction cost item.

### Construction Costs

The **Construction Costs** section of the **Cost** tab consists of the following two components:

- **Construction Costs Table** - This table allows you to specify an unlimited number of construction costs for each element.
- **Advanced Construction Costs Options** - This button is only available for elements such as pipes, inlets, gravity junctions, and manholes that support Unit Cost Functions. Clicking this button accesses advanced options for the selected construction cost item.

#### *Construction Costs Table*

The construction costs table allows you to specify an unlimited number of construction cost items for each element. Each construction cost item is composed of four basic characteristics which are listed below.

- **Label** - This is a string that identifies the construction cost item. It must be unique for every construction cost in the table.
- **Quantity** - This field holds a numeric value that will be multiplied by the unit cost to compute the total cost for the construction cost item.
- **Unit** - The value in this field signifies the unit of the value held in the quantity field. For pipes, this field can be either a length unit or "each." For nodal elements this field is a user-defined string.
- **Unit Cost** - The is the cost per unit specified in the unit column. For instance, for a pipe it could be cost per length. This value is multiplied by the quantity to calculate the total cost for the construction cost item. If a **Unit Cost Function** is assigned to a construction cost item then this field will not be editable as the value will be computed based on the **Unit Cost Function**.
- **Total Cost** - This is calculated by multiplying the unit cost by the quantity. The value in this field is always calculated by the program.

#### *Advanced Construction Cost Options*

Construction cost items for pipes and gravity structures (inlets, manholes, and junction chambers) have a set of advanced options. Under these advanced options, you can specify a **Unit Cost Function** to associate with a construction cost item. A **Unit Cost Function** describes the relationship between the unit cost for a construction cost item and the value of an attribute of the element. For instance, the unit cost for a pipe may be a function of the diameter. If you assign a **Unit Cost Function** to a construction cost item then the unit cost for that item is automatically updated as the physical characteristics of the element change.

For pipes there is an additional advanced option **Set Quantity Equal to Pipe Length**, which allows you to set the quantity field for a construction cost item equal to the length of the pipe.

## 6.3.11 User Data Tab

The **User Data** tab allows you to view and edit the customizable user data for each element. This tab is composed of two sections:

- **User Data** - Any Date/Time, Number, Text, and Yes/No data defined by the user.
- **User Memos** - Any memo data fields defined by the user.

For information on how to add new fields or edit an existing field format, see the Help on the **User Data Extension** dialog.

**Default user-defined attributes are provided. These can easily be deleted or modified.**

**User Data Extensions are a powerful way to add your own data to the project. This data will not affect the hydraulic calculations in any way, but can be used as any other data for operations such as sorting, annotating, reporting, and importing/exporting.**

#### User Data Section

This section contains a list of Date/Time, Number, Text, and Yes/No user data fields, displayed as single line fields. User data fields are defined in the **User Data Extension** dialog.

#### User Memos Section

This section contains a list of any memo fields displayed as multiple line scrolling text panes. User memos are defined in the **User Data Extension** dialog.

### 6.3.12 Message Tab

All **Element Editors** have a **Messages** tab, which contains three parts:

- **Message List** - Contains information that is generated during the calculation of the model, such as warnings, errors, and status updates.
- **Description** - An informative statement that you may enter about the element.
- **Notes** - Contains notes that you enter, and may include a description of the element, a summary of your data sources, or any other information of interest.

**Messages, descriptions, and notes will be printed in any element report.**

## 6.4 Loading Dialogs

### 6.4.1 Add New Load Dialog

Whenever an **Add** button is clicked under the **Loading** tab or when adding loads from within the **Sanitary (Dry Weather) Loading** and **Infiltration and Inflow Loading** alternatives, the **Add New Load** dialog will open up allowing you to select an appropriate load type to add as a load to the current element.

Select from the following load definitions:

- **Hydrograph** - Flow vs. Time
- **Unit Load** - Unit Type & Count (sanitary (dry-weather) loads only)
- **Pattern Load** - Base Flow & Pattern

When the **OK** button is clicked, a dialog will open where data associated with the selected load definition can be entered.

### 6.4.2 Base Load Dialog

- **Unit Sanitary (Dry Weather) Load** - Unit sanitary loads define the type of the load (for instance Apartment or Airport). Unit sanitary loads are selected, edited, and created in a **Unit Sanitary (Dry Weather) Load Library**, which is accessed by clicking the **ellipsis (…)** button next to this field.
- **Sanitary Unit Load Units** - Represents the local count of loading units for the selected unit sanitary load.
- **Loading Unit** - Individual entity inside the unit sanitary load (for instance Resident or Passenger) generating sewage.
- **Unit Load** - Sewage flow generated by one loading unit.
- **Base Load** - Average local sanitary load resulting from the category chosen under the **Unit Sanitary (Dry Weather) Load** column. This value is computed by multiplying the loading unit count by the unit load for each unit sanitary load and summing them up.

**Base load represents the average sanitary loads to the system. During a Steady-State analysis actual design sanitary loads are calculated using Extreme Flow Factor methods such as variable peaking methods, which account for various effects like system routing. During an Extended Period Simulation patterns can be applied to specific unit sanitary loads to describe how the load varies over time.**

### 6.4.3 Hydrograph Dialog

Hydrograph information is entered through this dialog.

- **Label** - A unique label that serves as an identifier for the set of hydrograph data.
- **Hydrograph** - In this table, the time / discharge data can be entered. The following functions can be performed on a row in this table:
  - **Insert** - will insert a new row above the selected row in the table.
  - **Duplicate -** will create a copy of the selected row.
  - **Delete -** will remove the selected row from the table.

**By clicking the Notes tab you can enter in relevant information about the hydrograph. You can also create a graph of the hydrograph by clicking the Plot button.**

### 6.4.4 Pattern Load Dialog

A Pattern Load consists of a single base load and a loading pattern that describes how that load varies over time for an Extended Period run.

If the pattern selected in the **Pattern** field is **Fixed** then the entered base load will remain constant over the entire Extended Period Simulation.

**During a Steady State analysis the base load is used as the load regardless of the pattern entered.**

# 6.5 Prototypes

Prototypes allow you to enter default values for the elements in your network. These default values are used while laying out the network. Prototypes can reduce data entry requirements dramatically if a group of network elements share common data. For example, if a section of the network contains all concrete pipes, use the pipe prototype to set the **Material** field to concrete. When a new pipe is created, its material attribute will default to concrete.

**Changes to the prototypes are not retroactive and will not affect any elements created prior to the change.**

**You can use the FlexTable Global Editing feature to change the data for any group of elements.**

**If a section of your system has distinctly different characteristics than the rest of the system, adjust your prototypes before laying out that section. This will save time when you edit the properties later.**

You can configure the element prototypes at the beginning of a new project during the **Project Setup Wizard**. You can also select **Tools / Prototypes** from the pull-down menu to edit the prototypes for the project at any time.

# 6.6 User Data Extensions

**User Data Extensions** are a set of one or more fields that you can define to hold data to be stored in the model. The User Data Extension feature allows you to add your own data fields to the project. For instance, you can add a field for keeping track of the date of installation for an element, or the type of area serviced by a particular element.

**User Data Extensions** exhibit the same characteristics as the pre-defined data used in and produced by the model calculations. This means that **User Data Extensions** can be imported or exported through database and Shapefile connections, viewed and edited in FlexTables, included in tabular reports or element detailed reports, annotated in the drawing, color coded, and reported in the detailed element reports. This data can also be accessed on the **User Data** tab of each **Element Editor** dialog.

**The user data does not affect the hydraulic model calculations. However, their behavior concerning capabilities like editing, annotating, sorting and database connections is identical to any of the standard pre-defined attributes.**

## 6.6.1 User Data Extensions Dialog

The **User Data Extension** dialog holds a summary of the user data extensions currently defined in the project. In this dialog, there is a tab for each type of element. By clicking a particular tab, you can access the user data extensions currently defined for that type of element. The software initially contains default user data extensions, but these can be deleted or edited. Each tab in the **User Data Extension** dialog is composed of a table listing characteristics of the user data extensions defined for that type of element. In addition, there are a series of buttons that can be used to add, edit, delete, and share individual user data extensions. The table listing the user data extensions consists of the following four columns:

- **Label -** Description that will appear next to the field for the user data extension, or as the column heading if the data extension is selected to appear in a FlexTable.

- **Type** - Lists the type of data that is valid for the data extension. The available data types are Date/Time, Number, Text, Memo, and Yes/No.
- **Unit / Picture** - Contains the unit of each numeric data extension, or the date and time presentation format for Date/Time data extensions. Both the unit and the date and time representations are specified when you create the data extension. They can always be modified by editing the data extension.
- **Shared** - If an asterisk appears in this column, it indicates that the user data extension is shared among two or more types of elements. See explanations on the **Existing Fields to Share With** dialog for more details.

The following list describes the four buttons that appear on the right side of the table:

- **Add** - Adds a new User Data Field. The **User Field Specification** dialog will open when you click this button. Here, you can define the properties of the user data extension that you are adding.
- **Edit** - You can edit an existing user data extension by highlighting the data extension you wish to edit and clicking this button. This will open the **User Field Specification** dialog where you can change the properties for that item.
- **Delete** - You can delete a data extension by highlighting it and clicking this button. If the data extension you are deleting is shared among multiple types of elements, it will only be removed from the element type that you are currently editing. If you remove a user data extension, all the information contained in that field will be permanently removed.
- **Share** - You can open the **Existing Fields to Share With** dialog by clicking this button. Here, you pick which of the available attributes defined for other types of elements you would like to share with the current type of element.

At the bottom of the **User Data Extension** dialog is a **File** button that allows you to import or save a set of user-defined data extensions. You can save the current configuration of user data extensions for later use by selecting **File / Save**, and specifying a file location and name. The file extension for the files holding the user data extension configurations is '.udx'. Select **File / Import** to merge the data extension configurations defined in these files into the current project. Importing a '.udx' file will not remove any of the other data extensions defined in your project. User data extensions that have the same name as those already defined in your project will not be imported.

To access the **User Data Extensions** dialog, click **Tools / User Data Extensions…** from the pull-down menus.

### User Field Specification Dialog

The properties defining a user data extension can be viewed and edited in the **User Field Specification** dialog, which is composed of two tabs:

**Type** - Enter the user data specification.

**Notes** - Enter any notes related to the User Data Specification.

#### *Type Tab*

The **Type** tab is composed of two sections:

**Type** - Contains fields for entering the label for the user data extension, as well as the data type.

**Format** - Contains fields for defining the specification of the type of user data extension selected in the **Type** section.

### *Type Section*

The **Type** section contains fields for entering the label and data type for the user data extension. The name entered in the **Label** field corresponds with the **User Data Extension** field on the **User Data** tab of the **Element Editor**. This label will also be used as the column heading if the user data extension is added to a FlexTable.

If you want the label to be displayed on multiple rows when it is used as a column heading, you can use forward slashes to specify the location of line breaks. When the label is used as a field label in a dialog, the forward slashes will be converted to spaces. In FlexTables, there is an option to use abbreviated labels for the column headings. If you want an alternative label to be displayed, you can specify an abbreviated label after the original label, and separate them by the bar symbol, '|'. When the option to display abbreviated labels is enabled in the FlexTables, this is the text that will be used as the column heading. For instance, if you specified the label **'Date/Installed | Date/Inst.'** it will be displayed in one of the following three ways, depending on the location and options selected.

| Field Label (Reports/Element Dialogs) | Column Heading (FlexTable) | Column heading with abbreviated label option selected (FlexTable) |
|---|---|---|
| **Date Installed** | **Date<br>Installed** | **Date<br>Inst.** |

You can select from five different types of data for the user data extension from the drop-down list in the **Type** field. An explanation of each is presented in the list below:

- **Date/Time** - Use this data type when you want the values you are entering to be in a standard date and time format. This format can be more useful than storing date information in a simple text field because it allows the dates to be sorted correctly when they appear in a FlexTable.
- **Memo** - If a user data extension is defined to be a memo, it will appear as a scrolling text pane in the **User Memos** section of the **User Data** tab in the **Element Editor** dialog.
- **Number -** Use this data type for fields that contain numeric values. You can specify a unit for the information in this field. The values contained in this field will then be automatically converted if you change the unit for this field.
- **Text** - Use this data type to create a single-line text field.
- **Yes/No** - Use this data type to display the attribute as a check box to represent true/false data.

### *Format Section*

This section is enabled only if you select Date/Time or Number in the **Type** section. Here is where you define the properties governing the type of data selected.

**Number Format** - If the type of data you selected was numeric, you can select a unit type (length, volume, intensity, etc.), a unit, a display precision, and whether to use scientific notation. There are no format options for memo, text, and Yes/No data types.

**Date/Time Type Format** - If you selected the **Date/Time** type, you can specify whether you would like the date or time to appear first in the input field, as well as the format of the date and time information. The format in which the date and time information will be displayed can either be selected from the drop-down lists, or you can type your own custom format directly into the **Date Picture** and **Time Picture** fields. If one of these fields is left blank, the corresponding information will not be displayed.

The **Date/Time** data type consists of an input and an output format. The input format is a fixed format that is determined by the regional settings on your computer. Whenever you enter information into a **Date/Time** field, the information must be entered according to the input format. If it is not entered in the proper input format, the value will simply revert to the original value.

The output format is simply a mask that defines the manner in which the date and time information will be displayed. It does not affect the way the date and time information can be entered into a **Date/Time** field.

The output format can be edited as follows:

- To specify dates with no leading zeros for single-digit days, years, or months, use lowercase d, lowercase y, or uppercase M.
- To specify dates with leading zeros for single-digit days, years, or months, use lowercase dd, lowercase yy, or uppercase MM.
- To specify abbreviations for the day, year, or month, use lowercase ddd, lowercase yyy, or uppercase MMM.
- To specify the full name of the day, year, or month, use lowercase dddd, lowercase yyyy, or uppercase MMMM.

If there are characters in the output format that do not map to valid date or time information, then the actual value of the character will be displayed. For example, if you wanted the date to be displayed as June 15, 1998, you would define the format as 'MMMM d, yyyy'. Since the spaces and comma do not map to any of the date information, their actual values are displayed. To include a piece of text that contains a character that maps to the date or time information, use single quotation marks (') around the text.

#### *Notes Tab*

This tab contains a text pane for entering notes about the current data extension. The text entered here is not displayed anywhere in the model, but allows you to keep records for a particular data extension.

### Existing Fields to Share With Dialog

This dialog allows you to choose which of the available attributes defined for other types of elements you want to share with the current type of element. The following sections are available:

- **Available Items** - Lists attributes defined for other element types that have not already been shared with the current type of element. In order to add attributes to the current element type, highlight them and click the **Add** button to transfer them to the **Selected Items** list.
- **Selected Items** - The attributes in the **Selected Items** list will be added to the current element after you click the **OK** button.

All the characteristics (such as data type, format, unit, and display precision) for a particular user data extension are the same for all the elements that share it. This is useful when the attribute you are adding needs to be the same for all the element types for which it is defined. For instance, if you have a **Date Installed** field for every element, sharing guarantees that the date format is the same for every element and will appear in a single FlexTable column. If, at a later point, you decide the date should be in a different format, you can change the format for one type of element. That change will filter through to all the elements that share that attribute.

# Chapter 7

# FlexTables

## 7.1 Tabular Reporting Overview

FlexTables provide you with a powerful data management tool that can be used to edit input data and present output data in a quick, efficient manner. Haestad Methods provides you with default element tables. However, these tables can be customized to fit your particular needs. You can also create your own tables combining various input and output data for different model elements. You can use FlexTables to view all elements in the network, all elements of a specific type (e.g. all pipes), or any subset of elements. Additionally, tables can be filtered, globally edited, and sorted to ease data input and present output data for specific elements.

FlexTables may also be used for creating results reports that can be sent to a printer, a file, the Windows clipboard, or copied to your favorite word processing and spreadsheet software.

## 7.2 Table Manager

The **Table Manager** provides support for creating, opening, and managing tables. Although the predefined tables provide access to most of the network element information, it is sometimes practical to present model results and input data through user-defined tables. The **Table Management** button provides the following tools for manipulating user-defined tables:

- **OK** - Open the selected table.
- **Close** - Exit the **Table Manager** dialog without opening a table.
- **Table Management / New** - Create a new table using the Create New Table and Table Setup dialogs.
- **Table Management / Edit** - Modify the layout of the selected table using the **Table Setup** dialog.
- **Table Management / Rename** - Rename the selected table.
- **Table Management / Duplicate** - Duplicate the selected table for additional customizing. This is a very useful feature when you need to change a predefined table.
- **Table Management / Delete** - Delete the selected table.
- **Table Management / Reset** - Reset a table's units to the current unit system or reset a predefined table to factory defaults.

**You cannot rename or delete the predefined tables that come with this software.**

**When you choose to print a table, the table name will be used as the title for the printed report. You can change the report title by renaming the table.**

To access the **Table Manager**, select the **Tabular Reports** button on the main toolbar, or choose **Report / Tables** from the pull-down menu.

### 7.2.1 Creating New Tables

To create a new table, open the **Table Manager** by clicking the **Tabular Reports** button on the main toolbar, or by choosing **Report / Tables** from the pull-down menu. In the **Table Manager** dialog, click the **Table Management** button and select **New**.

1. Specify the Table Type to indicate the type of network elements you want to display in your table.
2. Specify either a one or two row display for your table (in SewerCAD or StormCAD).
3. Enter the name of your new table in the **Enter the description for this table:** field. This name will also be used as the report title when this table is printed.
4. Click **OK** to accept these settings and proceed to the **Table Setup** dialog where you can define your table.

### 7.2.2 Two Row Tables

Two-row tables allow you to present more information in fewer columns by pairing up attributes, such as Upstream and Downstream Node for pipes, or Ground and Sump Elevation for nodes. The DOT Report is an example of a two-row table. You can only specify the number of rows when you create the table. You cannot change the number of rows in an existing table.

**If you choose to use a two-row table, only the attributes that can be represented in pairs will actually make use of two rows. Attributes that cannot be represented in pairs will only make use of the first row. The cell in the second row will be empty.**

**Global Edit is not available in two-row tables.**

### 7.2.3 Editing Tables

The **Edit** option allows you to modify the list of attributes that will appear in your table.

### 7.2.4 Duplicating Tables

The **Duplicate** option allows you to create a new table based on an existing table.

### 7.2.5 Deleting Tables

The **Delete** option allows you to delete any table that you have defined. You cannot delete the predefined tables.

### 7.2.6 Renaming Tables

The **Rename** option allows you to change the name of any table that you have defined. You cannot rename any of the predefined tables.

**The table name will be used as the title in printed reports. You cannot rename any of the predefined tables. If you need to rename a predefined table, duplicate it first and then rename it.**

### 7.2.7 Resetting Tables

**Reset Units to the Current Unit System** - This option is only available for tables that are in Local Units mode. Local Units mode allows the table to maintain its own "local" set of column properties, such as units and precision. Use this option to reset all units in the selected table to the defaults for the current unit system, which refers to the units used in the current project. You will be prompted to confirm before this action is performed.

**Reset to Factory Defaults** - You can reset any of the predefined tables to the factory defaults. This option is not available for tables that you create.

To reset units to the current unit system select **Table Management / Reset / Reset Units to <the current Unit System>** in the **Table Manager**.

To reset the table to factory defaults select **Table Management / Reset / Reset to Factory Defaults…** in the **Table Manager**.

## 7.3 Table Setup Dialog

The **Table Setup** dialog allows you to customize any table through the following options:

- **Table Type** - Allows you to specify the type of network elements that will appear in the table. For example, only pipes will appear in a "pipe" table.
- **Available Columns** - Contains all the attributes that are available for your table design, and will change based on the **Table Type** field.
- **Pick Button** - You can click on this button to access the categorized Quick Attribute Selector for selecting columns to be added to the tabular report. The selected column will be highlighted in the **Available Columns Window** to be easily added to the **Selected Columns** as seen fit.
- **Selected Columns** - Contains attributes that will appear in your custom designed table. When you open the table, the selected attributes will appear as columns in the table in the same order that they appear in the list. You can drag and drop or use the up and down buttons to change the order of the attributes in the table.
- **Allow Duplicate Columns** - An advanced feature that allows you to place two identical columns in the same table and set them to different unit systems.
- **Column manipulation buttons** - Allows you to select or deselect columns to be used in the table, as well as to arrange the order in which the columns will appear.

**The number next to the Selected Columns label indicates the number of columns that will appear in your table.**

To access **Table Setup** from the **Table Manager**, highlight the desired table and select **Edit** from the **Table Management** menu button.

### 7.3.1 Table Type

The **Table Type** field allows you to specify the types of elements that will appear in the table. It also provides a filter for the attributes that appear in the **Available Columns** list. When you choose a table type, the available list will only contain attributes that can be used for that table type. For example, only pipe attributes will be available for a "pipe" table.

### 7.3.2 Available Table Columns

The **Available Columns** list is located on the left side of the **Table Setup** dialog. This list contains all of the attributes that are available for the type of table you are creating. The attributes displayed in yellow represent non-editable attributes, while those displayed in white represent editable attributes.

### 7.3.3 Selected Table Columns

The **Selected Columns** list is located on the right-hand side of the **Table Setup** dialog. The attributes in this list will appear as columns in the table when it is opened. The columns will appear in the same order as the attributes in the selected list.

To add columns to the **Selected Columns** list:

1. Select one or more attributes in the **Available Columns** list.
2. Click the Add button [>] or drag and drop the highlighted attributes to the **Selected Columns** list.

### 7.3.4 Table Manipulation Buttons

The **Add** and **Remove** buttons are located in the center of the **Table Setup** dialog.

**[ > ]** Adds the selected item(s) from the **Available Columns** list to the **Selected Columns** list.

**[ >> ]** Adds all of the items in the **Available Columns** list to the **Selected Columns** list.

**[ < ]** Removes the selected item(s) from the **Selected Columns** list.

**[ << ]** Removes all items from the **Selected Columns** list.

To rearrange the order of the attributes in the **Selected Columns** list:

1. Highlight the item to be moved.
2. Move it up or down in the list by clicking the up or down button located below the Selected Columns list, or by simply dragging it to the desired location.

**You can select multiple attributes in the Available Columns list by holding down the Shift key or the Control key while clicking with the mouse. Holding down the Shift key will provide group selection behavior. Holding down the Control key will provide single element selection behavior.**

**The items displayed in yellow represent non-editable columns (e.g. columns that contain calculated data) while those in white represent editable columns (e.g. columns that contain input data).**

### 7.3.5 Allow Duplicate Columns

Set this check box to allow duplicate columns in a table. **Allow Duplicate Columns** is an advanced feature that allows you to place two identical columns in the same table and set them to different unit systems.

## 7.4 Table Window

The **Table** window is where you will perform most of your data input and review. It has many features to assist you with data entry, data formatting, report customization, and output generation. To access the **Table** window, highlight a table in the **Table Manager**, and click **OK**. Here are some of the topics that will be covered in this section:

- Table Navigation
- Table Customization

Options:

- Sorting Tables
- Filtering Tables
- Changing Column Headings
- Globally Editing Data
- Local vs. Synchronized Units
- Mixing Units in a Tabular Report
- Abbreviated Labels
- Changing Column Display Properties

Output:

- File (Export Table to ASCII File)
- Table Copy to Clipboard
- Table Print
- Table Print Preview

Columns:

See the Glossary for information regarding the definitions of the columns in the **Table** window.

> **Use the Scenario control located at the top of the Table Window to quickly view the data for different scenarios.**

To access the **Table** window, open the **Table Manager**, highlight the table you wish to open, and click **OK**.

## 7.4.1 Editing Tables

### Editable Table Columns

Editable table columns correspond to input data that you can change. The values in these columns can be modified either directly or through the **Global Edit** option. These columns are displayed with a white background.

Non-editable table columns are displayed with a yellow background, and correspond to model results calculated by the program and composite values.

### Table Navigation

The **Table** window supports two modes: Table Navigation Mode and Cell Navigation Mode. By pressing the **F2** key, you can toggle between them.

#### *Table Navigation Mode*

The Arrow keys, Home, End, PgUp, PgDn, Ctrl+<arrow> keys navigate to different cells in a table. Table Navigation Mode is the default mode when editing a table. To edit within a single cell of a table, press the **F2** key to switch to Cell Navigation Mode.

#### *Cell Navigation Mode (Edit Mode)*

In Cell Navigation Mode, the Arrow keys, Home, and End keys navigate within a single cell. When Cell Navigation Mode is active, the word "EDIT" will appear on the status pane at the bottom of the window. Cell Navigation Mode will automatically terminate when you press any key except for Left, Right, Home, End, Delete, or Backspace.

#### Globally Editing Data

You can globally change the values of any editable column in a table. Right-click the column that you wish to globally change and choose the **Global Edit** menu item.

For numeric columns:

1. Choose the operation to be performed: Add, Divide, Multiply, Set, or Subtract.
2. Enter the value you wish to use.
3. Click **OK**, and the values in the entire column will be updated to reflect this change.

For non-numeric columns:

1. Enter the new value.
2. Click **OK**, and the values in the entire column will be updated to reflect this change.

> **Global Edit is available only for editable columns. Global Edit is not available in 2-row tables (in SewerCAD or StormCAD). You can use Global Edit in conjunction with Filtering to globally edit a subset of elements.**

### 7.4.2 Sorting/Filtering Tables

#### Sorting Tables

Tables can be sorted based on a single column, multiple columns, or network order.

#### Sort By Network

In a network-based sort, elements in the table will be sorted so that structures furthest from the outlet will appear at the top of the list, and structures closest to the outlet will appear at the bottom of the list.

> **"Furthest from the outlet" as used here does not refer to an actual distance, but to the number of pipes that are between the specified element and the outlet. If your system contains multiple networks, the elements in the first network will be sorted as described above, followed by the elements in the second network, and so on.**

## Custom Sort

You can sort elements in the table based on one or more columns, in ascending or descending order. For example, the following table is given:

| Slope (ft/ft) | Depth (ft) | Discharge (cfs) |
|---|---|---|
| 0.001 | 1 | 4.11 |
| 0.002 | 1 | 5.81 |
| 0.003 | 1 | 7.12 |
| 0.001 | 2 | 13.43 |
| 0.002 | 2 | 19.00 |
| 0.003 | 2 | 23.27 |

A custom sort is set up to sort first by Slope, then by Depth, in ascending order. The resulting table would appear in the following order:

| Slope (ft/ft) | Depth (ft) | Discharge (cfs) |
|---|---|---|
| 0.001 | 1 | 4.11 |
| 0.001 | 2 | 13.43 |
| 0.002 | 1 | 5.81 |
| 0.002 | 2 | 19.00 |
| 0.003 | 1 | 7.12 |
| 0.003 | 2 | 23.27 |

To access the custom sort capability, open the table you wish to sort. Then, right-click a column label and select **Sort / Custom**.

## Filtering Tables

To access the filtering operations, use the **Options** button at the top of the **Table** window (in the case of a FlexTable), or right-click the column header by which you wish to filter. Filters allow you to change the table so only rows that match the specified criteria will appear.

- **Quick Filter** - Set up a simple filter by right-clicking the column by which you wish to filter.
- **Custom Filter** - Set up a custom filter based on one or more criterion.
- **Reset** - Turn off the active filter, causing all available rows in the table to be displayed.

**Another way to select which elements are displayed in the table consists of first selecting elements, either graphically, or by using the Selection Set tool. Then, right-click any of the selected elements and choose Edit Group from the pop-up menu that appears. This will display the Table Manager dialog. Only the selected elements will appear in any of the tables you open at this point.**

When you perform a **Quick Filter** or a **Custom Filter**, the **Filter** dialog will open allowing you to specify your filtering criterion.

Each filter criterion is made up of three items:

- **Column -** The attribute to filter.
- **Operator -** The operator to use when comparing the filter value against the data in the specific column (operators include: =, >, >=, <, <=, <>).
- **Value -** The comparison value.

Any number of criterion elements can be added to a filter. Multiple filter criterion are implicitly joined with a logical "AND" statement. When multiple filter criterion are defined, only rows that meet all of the specified criteria will be displayed. A filter will remain active for the associated table until the filter is reset.

The status pane at the bottom of the **Table** window always shows the number of rows displayed and the total number of rows available (e.g. "10 of 20 elements displayed"). When a filter is active, this message will appear in a highlighted color.

Table filtering allows you to perform global editing on any subset of elements. Only the elements that appear in the filtered table will be edited.

### 7.4.3 Table Customization

There are several ways to customize tables to meet a variety of output requirements:

- **Changing the Report Title** - When you print a table, the table name is used as the title for the printed report. You can change the title that appears on your printed report by renaming the table, using the **Table Manager**.
- **Adding/Removing Columns** - You can add, remove, and change the order of columns by using the **Table Setup** dialog. Use the **Table Manager** to access the **Table Setup** dialog.
- **Drag/Drop Column Placement** - With the **Table** window open, select the column that you would like to move by holding down the left mouse button on its column heading. Drag the column heading to the left or right, and release the mouse button to drop the column into its new location.
- **Resizing Columns** - With the **Table** window open, place your pointer over the vertical separator line between column headings. Notice that the cursor changes shape to indicate that you can resize. Hold down the left mouse button and drag the mouse to the left or right to "stretch" the column to its new size. When you are satisfied, release the mouse button to set the new column width.
- **Changing Column Display Properties** - With the **Table** window open, right-click in the heading area of the column you wish to change and choose the **Properties** menu item. The current column properties will be displayed in the **Set Field Options** dialog. Refer to the section on Local Units for additional information.
- **Changing Column Headings** - With the **Table** window open, right-click the column heading that you wish to change and choose **Edit Column Label**. Refer to section on Changing Column Headings for additional information.

#### Changing Column Headings

To change the label of any column in the **Table** window, right-click the column heading that you wish to change and choose **Edit Column Label** from the context menu. The backslash character (\) can be used to insert a line-break wherever you want the title to be split into multiple lines. If you enter an empty label, the column heading will be restored to the default label.

#### Abbreviated Labels

Using label abbreviations will allow columns to take up less space. This will permit more data to fit on each page when printing a report. If you wish to define an abbreviated label, right-click the desired column heading and choose **Edit Column Label**. In the **Label** dialog, separate the abbreviated label from the default label with the '|' symbol, located above the backslash (\) on most keyboards. For example, to use the abbreviation **L** for the **Length** column, type **Length|L** in the field provided. When the **Use Label Abbreviations** option is toggled on, the abbreviated label will appear.

To toggle the **Use Label Abbreviations** option on and off, select **Options / Use Abbreviated Labels** from the **Table** window.

## Changing Column Display Properties

You can change the display properties (e.g. units, precision) of any numeric column in the **Table** window. Right-click the label of the column that you wish to change, and select **Properties** from the pop-up menu. This opens the **Set Field Options** dialog, where you can change the display properties of the column.

## Local vs Synchronized Units

Use the **Options** button at the top of the **Table** window to access the **Use Local Units** menu item. Click the menu item to toggle between **Local Units** and **Synchronized Units**. A check mark will appear next to the **Use Local Units** menu item to indicate that Local Units mode is active. Otherwise, Synchronized Units mode is active.

- **Synchronized Units** - This is the default mode that allows the table to stay synchronized with the active project. If you have one project in US Customary and one project in SI units, the Table will match the units in the project that is currently open.
- **Local Units** - Local Units mode allows the table to maintain its own "local" set of column properties (units, precision, etc). This is a powerful feature that gives you the ability to build tables that are always in a fixed unit system, no matter what unit system the active project is currently using. This is useful for printing reports in different unit systems.

When the **Table** window is open, the current unit synchronization mode is displayed in the status pane at the bottom of the window.

## Mixing Units in a Tabular Report

This software allows for duplicate columns in a table, thus giving you the ability to display an attribute in different units.

For example, to see two "Pipe Length" columns in a Table, one in feet and one in meters:

1. Open the Table Manager.
2. Click the **Table Management** button, and select **New** to create a new table.
3. Select the **Pipe Table Type** from the choice list, and enter a name for your new table. Click **OK** and you will be taken to the **Table Setup** dialog where you can customize your table.
4. In the **Table Setup** dialog, activate the **Allow Duplicate Columns** check box located at the lower left corner of the dialog.
5. Add the **Length** column to the **Selected Columns** list.

> **The Length column will still appear in the Available Columns list, but will be displayed in a lighter color, indicating that it has already been selected.**

6. Add the **Length** column again.
7. Click **OK** to close the **Table Setup** dialog. From the **Table Manager**, highlight the table you have just created, and click **OK**.
8. Click the **Options** button at the top of the window and select the **Use Local Units** menu item to turn Local Units on. You will be prompted to verify that you want to use local units. Click **Yes**.

9. Right-click the first **Length** column and select **Length Properties** to set the units in the column to "ft." Then, right-click the second **Length** column to set the units to "m."

## 7.4.4 Table Output

### Table Copy to Clipboard

The **Copy** button at the top of the **Table** window allows you to copy tab delimited data to the Windows clipboard. Tab delimited data can be pasted directly into your favorite spreadsheet program or word processor.

### Table Print

The **Print** button at the top of the **Table** window is used to output the table directly to the printer.

### Table Print Preview

Click the **Print Preview** button at the top of the **Table** window to view the report in the format that will be printed.

**Using label abbreviations will allow some columns to be narrower, permitting more data to fit on each page. Use the Options button at the top of the Table window to access this option.**

**Printing with landscape orientation will also allow more columns to fit on a single page. From the Print Preview window, use the Options / Print Setup menu item to access orientation.**

**Clicking the Copy button will allow you to paste the formatted information into another application such as Microsoft Word.**

### File (Export Table to ASCII File)

You may export the data shown in the **Table** window to an ASCII text file in either tab or comma-delimited format.

To export a table to an ASCII File format, select **File / Export Data** and either **Tab Delimited** or **Comma Delimited** from the **Table** window.

# Chapter 8

# Scenarios and Alternatives

## 8.1 Overview

The scenario management feature allows you to easily analyze and recall an unlimited number of "*What If?*" calculations for your model. The powerful two-level design, which uses Scenarios that contain Alternatives, gives you precise control over changes to the model, while eliminating any need to input or maintain redundant data.

We have worked hard to devise a system that offers the power and flexibility that you demand, with the ease of use that you have come to expect from us. If you are like most users, you will want to jump right in without having to spend a lot of time reading. When you are ready to create your first scenario, you will find that you will be able to accomplish what you want easily and quickly.

The **Scenario Wizard** is designed to get you started quickly, while slowly exposing you to the power behind scenarios and alternatives.

When you are ready to model more complex scenarios, you will appreciate the power and flexibility provided by the various scenario management features.

If you are a beginning user, try the **Scenario Wizard** and run the Scenario tutorial. Also, refer to the Scenario Management Reference Guide in Appendix D. Be sure to read about Alternatives, and investigate the **Alternatives Manager** dialog.

## 8.2 Alternatives

Alternatives are the building blocks behind scenarios. They are categorized data sets that create scenarios when placed together. Alternatives hold the input data in the form of records. A given record holds the data for a particular element in your system. The different types of alternatives are as follows:

- Physical Properties
- Sanitary (Dry Weather) Loading
- Infiltration and Inflow Loading
- Known Flow Loading
- Structure Headlosses
- Boundary Conditions
- Design Constraints
- Initial Settings
- Operational
- Cost
- User Data

The exact properties of each alternative are discussed in their respective sections. By breaking up alternatives into these different types, we give you the ability to mix and match different alternatives within any given scenario.

Scenarios are composed of alternatives, as well as other calculation options, allowing you to compute and compare the results of various changes to your system. Alternatives can vary independently within scenarios, and can be shared between scenarios.

There are two kinds of alternatives: Base alternatives and Child alternatives. Base alternatives contain local data for all elements in your system. Child alternatives inherit data from base alternatives, or even other child alternatives, and can contain data for one or more elements in your system. The data within a child alternative consists of data inherited from its parent and the data altered specifically by you (local data).

When you first set up your system, the data that you enter is stored in the various base alternatives. If you wish to see how your system will behave, for example, by increasing the diameter of a few select pipes, you can create a child alternative to accomplish that. You can make another child alternative with even larger diameters, and another with smaller diameters. There is no limit to the number of alternatives that you can create.

Scenarios allow you to specify the alternatives you wish to analyze. Once you have determined an alternative that works best for your system, you can permanently merge changes from the preferred alternative to the base alternative if you wish.

Remember that all data inherited from the base alternative will be changed when the base alternative changes. Only local data specific to a child alternative will remain unchanged.

### 8.2.1 Alternatives Manager

The **Alternatives Manager** is a central location for managing all of the alternatives in your project.

Across the top of the alternative manager are tabs for each type of alternative. The tabs access a tree view of the available alternatives of the following tabs:

- Physical Properties
- Sanitary (Dry Weather) Loading
- Infiltration and Inflow Loading
- Known Flow Loading
- Structure Headlosses
- Boundary Conditions
- Design Constraints
- Initial Settings
- Operational
- Cost
- User Data

On the right side of the dialog are a number of buttons that provide functions for managing the alternatives. The following list provides a brief description of the function of each of these buttons:

- **Add** - Create a new base alternative, first prompting for a name, then opening an **Alternatives Editor**. Base alternatives are initialized with the data that is currently entered either in tables or specific element dialogs.

- **Add Child** - Create a new child alternative that inherits data from the selected parent alternative. This allows you to automatically share the majority of the records with a parent alternative, while modifying only selected records in the child alternative.
- **Edit** - Open the tabular record editor for the selected alternative. This tabular record contains the values that are used by the selected alternative.
- **Merge** - Move all records from the selected child alternative into its parent alternative, and then remove the selected alternative. The records in the selected alternative will replace the corresponding records in the parent. This is helpful when you have been experimenting with changes in a child alternative, and you want to permanently apply those changes to the parent alternative. All other alternatives that inherit data from that parent alternative will reflect these changes.
- **Rename** - Rename an existing alternative. This invokes an in-place editor in the tree view of the available alternatives. Make the desired changes to the existing name, and press the **Enter** key
- **Duplicate** - Create a new alternative filled with records copied from the selected alternative. Use this if you wish to copy the data from an alternative but not create a child. The two alternatives will be independent.
- **Delete** - Remove the selected alternative and its records. Deleting an alternative will also delete all of the input data associated with that alternative. You cannot delete an alternative that has children associated with it.
- **Report** - Generate a Print Preview of a summary report of the selected alternative, all alternatives, or the selected alternative and all of its children in that hierarchy.

**You will not be allowed to merge or delete an alternative that is referenced by one or more scenarios. When you attempt to perform the operation, you will be provided with a list of the scenarios that reference the alternative.**

**If you are attempting to merge an alternative that is referenced, you will need to edit the scenario(s) that references the alternative you are merging, and make them reference the parent alternative to which you are merging. Use the Scenario Manager window to edit the scenario(s), and the Alternatives tab to make the scenario point to the parent alternative.**

### 8.2.2 Alternatives Editor

The **Alternatives Editor** displays all of the records held by a single alternative. These records contain the values that are active when a scenario referencing this alternative is active. They allow you to view all of the changes that you have made for a single alternative. They also allow you to eliminate changes that you no longer need.

There is one editor for each alternative type. Each type of editor works basically the same, and allows you to make changes to a different aspect of your system. The first column contains check boxes, which indicate the records that have been changed in this alternative.

- If the box is *checked*, the record on that line has been modified and the data is local, or specific, to this alternative.
- If the box is *not checked*, it means that the record on that line is inherited from its higher-level parent alternative. Inherited records are dynamic. If the record is changed in the parent, the change will be reflected in the child. The records on these rows reflect the corresponding values in the alternative's parent.

**As you make changes to records, the check box will automatically become checked. If you want to reset a record to its parent's values, simply uncheck the corresponding check box.**

**Many columns support Global Editing, allowing you to change all values in a single column. Right-click a column header to access the Global Edit option.**

**The checkbox column will be disabled when you edit a base alternative.**

To access the **Alternatives Editor** for a particular alternative, select **Analysis\Alternatives** from the pull-down menu to activate the **Alternatives Manager**. Select the desired alternative tab, choose the alternative you would like to edit, and click the **Edit** button.

## 8.2.3 Physical Properties Alternative Editor

The **Physical Properties Alternative Editor** allows you to modify the physical properties of any network element, including gravity pipes, pressure pipes, manholes, junction chambers, pressure junctions, pumps, and wet wells.

### Physical Properties Alternative Editor for Gravity Pipes

The **Physical Properties Alternative Editor** for gravity pipes is used to create various data sets for the physical characteristics of gravity pipes. The following properties are available for editing:

- **Set Invert to Upstream Structure** - Specifies whether if the invert will be entered by the user or calculated using upstream structure sump elevation and desired sump depth.
- **Upstream Invert Elevation** - Elevation for the upstream invert.
- **Set Invert to Downstream Structure** - Specifies whether if the invert will be entered by the user or calculated using downstream structure sump elevation and desired sump depth.
- **Downstream Invert Elevation** - Elevation for the downstream invert.
- **Section Shape** - Shape of a pipe.
- **Material** - Type of material of which a pipe is made. The available pipe materials are contained in the **Material Engineering Library**. The **Material Library** allows you to create new materials and customize existing ones.
- **Roughness Coefficient** - Specify a roughness coefficient for a pipe based on the pipe material selected, or enter a unique value.
- **Section Size** - Specify new dimensions for a particular pipe segment. The available dimensions for a particular section shape are contained in the **Section Size Library**. You may add new values to this library if the desired size is not available.
- **Number of Sections** - Specify the number of pipe sections.
- **User-Defined Bend Angle** - Choose to either enter a bend angle or accept the bend angle calculated by the program from the graphical layout.
- **Bend Angle** - Specifies the bend angle of the pipe. If the **User Defined Bend Angle** box is checked, the field is editable.

### Physical Properties Alternative Editor for Pressure Pipes

The **Physical Properties Alternative Editor** for pressure pipes is used to create various data sets for the physical characteristics of pressure pipes. The following properties are available:

- **Upstream Invert Elevation** - Elevation for the upstream invert.
- **Downstream Invert Elevation** - Elevation for the downstream invert.
- **Set Invert to Downstream Structure** - Specifies whether if the invert will be entered by the user or calculated using downstream structure sump elevation and desired sump depth.
- **Material** - Type of material of which a pipe segment is made. The available pipe materials are contained in the **Material Engineering Library**. The **Material Library** allows you to create new materials and customize existing ones.
- **Diameter** - Inside diameter of the pipe.
- **Roughness** - Measure of the pipes' internal roughness based on the pipe material selected, or enter a unique value.
- **Minor Loss Coefficient** - Appurtenances such as valves, bends, and tees contribute to local flow disturbances resulting in energy loss. Minor losses can be entered in one of three ways:
  - Manually enter a value for the minor loss.
  - Select a minor loss from the existing list. When you click the minor loss field, a choice list will appear. Click this to access the Minor Loss Coefficients that are stored in the **Minor Loss Library**.
  - Create a composite minor loss. When you click a minor loss field, an **ellipsis (…)** button will appear. Click this to access the **Minor Loss Elements Editor**. Here you can specify the quantity of a particular type of minor loss. If you need a minor loss that is not available, click the **ellipsis (…)** button to access the **Minor Loss Library**. You can edit existing minor losses or create new ones. Use the **Minor Loss Elements Editor** to enter as many types of minor losses as you need.
- **Check Valve** - A check box you can mark to indicate the presence of a check valve.

## Physical Properties Alternative Editor for Manholes

The **Physical Properties Alternative Editor** for manholes is used to create various data sets for the physical characteristics of manholes. The following properties are available:

- **Ground Elevation** - Ground elevation at the manhole.
- **Sump Elevation** - Elevation of the manhole sump.
- **Set Rim Equal to Ground Elevation** - Specify whether the ground elevation should be set equal to the rim elevation of the manhole. If the box is checked, the rim elevation will be automatically set equal to the ground elevation. Otherwise, the rim elevation will be user-defined.
- **Rim Elevation** - Specify the elevation of the rim if the box in the column labeled **Set Rim Equal to Ground Elevation** is not checked. If it is checked, the rim elevation is automatically set equal to the ground elevation.
- **Bolted Cover** - Specify whether the manhole cover is bolted. If the cover is not bolted and the Hydraulic Grade is determined to be above the rim or ground elevation, whichever is higher, the model will report a surcharged condition and the Hydraulic Grade will be reset to the higher of the rim and the ground elevation for the next calculation. If the cover is bolted and the Hydraulic Grade is determined to be above the ground or rim elevation, it will NOT be reset.
- **Has Diversion** - Determines if this element diverts flows out of the network or not. If this box is checked, the user will be able to edit diversion properties.
- **Diversion Target** - Determines the destination for diverted flows. To edit this field, the **Has Diversion** box must be checked.

- **Diversion Rating Table** - Editor button for Diversion Rating Curve Table. To edit this field, the **Has Diversion** box must be checked.
- **Structure Diameter** - Diameter of the manhole structure.

### Physical Properties Alternative Editor for Junction Chambers

The **Physical Properties Alternative Editor** for junction chambers is used to create various data sets for the physical characteristics of junction chambers. The following properties are available:

- **Ground Elevation** - Ground elevation at a junction chamber.
- **Top Elevation** - Top elevation of a junction chamber.
- **Bottom Elevation** - Bottom elevation of a junction chamber.
- **Has Diversion** - Determines if this element diverts flows out of the network or not. If this box is checked, the user will be able to edit diversion properties.
- **Diversion Target** - Determines the destination for diverted flows. To edit this field, the **Has Diversion** box must be checked.
- **Diversion Rating Table** - Editor button for Diversion Rating Curve Table. To edit this field, the **Has Diversion** box must be checked.
- **Structure Diameter** - Diameter of the junction structure. For non-circular junction chambers, an equivalent diameter should be entered.

### Physical Properties Alternative Editor for Wet Wells

The **Physical Properties Alternative Editor** for wet wells is used to create various data sets for the physical characteristics of wet wells. The following properties are available:

- **Ground Elevation** - Ground elevation at a wet well.
- **Base Elevation** - Elevation of the base of the wet well.
- **Minimum Elevation** - Lowest possible water surface elevation for the wet well. This elevation must be above the base elevation.
- **Maximum Elevation** - Highest possible water surface elevation for the wet well.
- **Section** - Physical parameters that define the wet well cross sectional geometry. There are two types of wet well sections, Constant Area and Variable Area. Click this field, then click the **ellipsis (…)** button that appears. This will activate a dialog allowing you to edit the parameters for the active wet well section type.

### Physical Properties Alternative Editor for Pumps

The **Physical Properties Alternative Editor** for pumps is used to create various data sets for the physical characteristics of pumps. The following properties are available:

- **Elevation** - Elevation of the center of the pump.
- **Pump Type** - Attributes that define the pump's operating characteristics. Click this field, then click the **ellipsis (…)** button that appears. This will activate a dialog allowing you to edit the parameters for the active pump type.

### Physical Properties Alternative Editor for Pressure Junctions

The **Physical Properties Alternative Editor** for pressure junctions is used to create various data sets for the physical characteristics of pressure junctions. The following property is available:

- **Ground Elevation** - Ground elevation at a pressure junction.

### Physical Properties Alternative Editor for Outlets

The **Physical Properties Alternative Editor** for outlets is used to create various data sets for the physical characteristics of outlets. The following properties are available:

- **Ground Elevation** - Ground elevation at the outlet.
- **Sump Elevation** - Elevation of the outlet sump.
- **Set Rim Equal to Ground Elevation** - Specify whether the ground elevation should be set equal to the rim elevation of the outlet. If the box is checked, the rim elevation will be automatically set equal to the ground elevation. Otherwise, the rim elevation will be user-defined.
- **Rim Elevation** - Specify the rim elevation if the box in the column **Set Rim Equal to Ground Elevation** is not checked. If the box is checked, the rim elevation is automatically set equal to the ground elevation.

## 8.2.4 Sanitary (Dry Weather) Loading Alternative Editor

The **Sanitary (Dry Weather) Loading Alternative Editor** allows you to edit the sanitary (dry weather) loads at manholes, pressure junctions, and wet wells. Sanitary loads, which comprise the base load of the sewer system, are loads produced by residential, commercial, recreational, and industrial activities.

The tabbed dialogs for manholes, pressure junctions, and wet wells all follow the same format.

The alternative contains the following columns:

- **Sanitary Load Type** - Identifies the type of load: **Base Load**, **Pattern Load**, or **Hydrograph** if there is a single load associated with the node. If there are multiple loads associated with the node then the **Load Description** will display **Composite**. By clicking a **Load Description** cell in the table, a button will be displayed. Click this button to access the **Composite Sanitary Load** dialog. From this dialog you can edit each of the individual loads.
- **Sanitary Unit Load Type** - Select the unit load type as established in the in the Unit Sanitary Load Engineering Library. If multiple sanitary unit loads are associated to the node then the field will display **Composite**.
- **Sanitary Unit Load Count** - Specify the number of loading units. If multiple sanitary unit loads are associated to the node then the field will display **N/A**.
- **Sanitary Unit Load Units** - Displays the loading units associated with the sanitary unit load type. For example, if the unit load type is "Apartment" the actual unit per a quantity of flow could be "Resident." If there is more than one unit loads associated with the elements the field will display **N/A**.
- **Sanitary Pattern Load Base Flow** - Enter a single load associated with the **Sanitary Pattern**. If multiple pattern loads are associated with the structure then this field will display the sum of all the sanitary base loads.
- **Sanitary Pattern Load Pattern** - Select the pattern, which describes how the **Sanitary Base Load** varies over time. If multiple pattern loads are associated with the node then the **Sanitary Pattern** field will display **Composite**. Click the **ellipses (…)** button in the field (if editable) to open up the **Pattern Manager**.

### Sanitary Load Composite Dialog

The **Sanitary Load Composite** dialog allows you to apply an unlimited number of different sanitary loads to a single node.

The following operations can be performed on the **Sanitary Flow** list in the section:

- **Add** - Add a new load to the **Sanitary Flow** list.

- **Edit** - Opens the editor for an existing load in the list.
- **Delete** - Deletes a sanitary load from the list.

The **Sanitary (Dry-Weather) Flow** section is specified as a collection of sanitary loads applied to the selected node. The following types of loads can be applied:

- **Hydrograph** - A flow vs. time distribution
- **Unit Load** - The type of **Unit Load** and the number of units associated with **Unit Load**. For example, 5000 passengers at an Airport.
- **Pattern Load** - A direct, known sanitary load with a set pattern.

### 8.2.5 Infiltration and Inflow Loading Alternative Editor

The **Infiltration and Inflow Loading Alternative Editor** allows you to edit the wet weather loads at gravity pipes, manholes, and wet wells. Wet weather loads are those produced by the introduction of rainfall water into the sewer system through groundwater infiltration or rainfall inflow.

Groundwater infiltration, which occurs along gravity pipes, is specified on the **Gravity Pipe** tab of the **Infiltration and Inflow Loading Alternative Editor**. Inflow, which occurs at manholes and wet wells, is specified on the **Inflow Structure** tab.

#### Infiltration and Inflow Loading Alternative Editor for Gravity Pipes

The **Infiltration and Inflow Loading Alternative Editor** for gravity pipes is used to specify the infiltration type and any additional infiltration for each gravity pipe individually. The following columns are available:

- **Infiltration Load Type** - Select how the direct infiltration along a gravity pipe will be computed. Click this field and a choice list will appear. Choose from the list of the available infiltration types. Once you have selected an infiltration type you can click the **ellipses (…)** button to bring up a dialog to edit data related to that infiltration type. For example, if **Hydrograph** is selected, clicking the **ellipses (…)** button will allow you input the associated flow vs. time data.
- **Infiltration Loading Unit** - Specify the unit used to define infiltration rates.
- **Infiltration Rate per Loading Unit** - Specify the rate of infiltration for the selected loading unit and infiltration type.
- **Infiltration Unit Count** - Editable when Count Based is selected as the Infiltration Type.
- **Base Flow** - Editable when Pattern Load is selected as the Infiltration Type.
- **Pattern** - Editable when **Pattern Load** is selected as the **Infiltration Type**. Click the **ellipses (…)** button in the field (if editable) to open up the **Pattern Manager**.
- **Infiltration Additional Flow** - Specify a lump sum infiltration amount for each gravity pipe. The total infiltration into a single gravity pipe is the sum of the direct additional infiltration.

#### Infiltration and Inflow Loading Alternative Editor for Inflow Structures

The **Infiltration and Inflow Loading Alternative Editor** for inflow structures is used to specify the list of wet weather inflow loads into manholes and wet wells. The following columns are available:

- **Inflow Load Type** - The column will display **Hydrograph** or **Pattern Loads** if there is a single load of that type associated with node. Otherwise, if multiple inflow loads are associated with the node, the column will display **Composite.** To access the **Composite Inflow** dialog, click a cell in the **Inflow Load Description** column. A button will appear. Click on this button to bring up the **Composite Inflow** dialog.

- **Inflow Pattern Load Base Load** - Enter a single load associated with the **Inflow Base Pattern**. If multiple pattern loads are associated with the structure then this field will display the sum of all the inflow base loads.
- **Inflow Patten Load Pattern** - Select the pattern, which describes how the **Inflow Base Load** varies over time. If multiple pattern loads are associated with the node then the **Inflow Pattern** field will display **Composite**. Click the **ellipses (…)** button in the field (if editable) to open up the **Pattern Manager**.

#### Composite Inflow Dialog

The **Composite Inflow** dialog allows you specify the Inflow loading for structure in the **Infiltration and Inflow Loading Alternative.**

The following operations can be performed on the **Inflow** list in the section:

- **Add** - Add a new load to the **Inflow** list.
- **Edit** - Opens the editor for an existing load in the list.
- **Delete** - Deletes an inflow load from the list.

The **Inflow** section is specified as a collection of wet weather loads applied to the selected node. The following types of loads can be applied.

- **Hydrograph** - A flow vs. time distribution
- **Pattern Load** - A direct, known wet weather load with a set pattern.

### 8.2.6 Known Flow Loading Alternative Editor

The **Known Flow Loading Alternative Editor** allows you to specify known flows at individual manholes and wet wells. Both the **Manhole** and **Wet Well** tabs are identical.

Known flows are a special type of fixed load. They remain constant as they progress downstream, and combine directly as a simple sum, similar to additional loads. During a Steady-State analysis when a known flow is specified at a downstream manhole or wet well, the local known flow replaces the upstream known flow rather than being added directly to the upstream known flow. During an Extended Period Simulation known flows generate constant load hydrographs which are additive. **Known Flows do not replace each other during an Extended Period Simulations**.

### 8.2.7 Structure Headlosses Alternative Editor

The **Structure Headlosses Alternative Editor** allows you to change the method used to calculate the headloss through each manhole and junction chamber. Both the manhole and the junction chamber tabs are identical. The available methods are:

- Absolute
- Standard
- HEC-22 Energy
- AASHTO
- Generic

For further information on these methods, see Appendix B.

## 8.2.8 Boundary Conditions Alternative Editor

The **Boundary Conditions Alternative Editor** gives you the ability to modify the tailwater condition and tailwater elevation at the outlets. You can specify free outfall, crown, or user-specified tailwater elevation. The program determines the tailwater elevation for all the options except user-specified.

## 8.2.9 Design Constraints Alternative Editor

The **Design Constraints Alternative Editor** allows you to edit the gravity pipe and gravity structure constraints governing the design of the system. It also allows you to specify which gravity elements you want designed, and the extent to which you want them designed. For example, you may want to design a particular pipe. However, you may only want to design the downstream invert elevation to meet a particular velocity, cover, and slope constraint. The **Design Constraints Alternative Editor** allows you to accomplish this.

The tabbed dialog for each particular type of element follows the same general format. The top of the dialog box contains several fields where the default design constraints can be entered. The constraints entered in these fields are applied to every element in the table on the bottom of the dialog, except the elements that are specified to contain local values. This system allows you to rapidly enter the values that govern most of the elements in the table, then manually override the constraints for those elements that are exceptions to the majority.

In order to specify that an element contains local data, place a check mark in the column labeled **Specify Local Constraints** on the same row as the element. The local design constraints, if specified, will override the default design constraints during an automatic design for that particular element. When the check mark appears, the yellow columns that display the global design constraints defined in the top of the dialog will turn white on the row of the element that is being modified. This means that you can now change the design constraint values for this particular element. If you click the check mark again, the opposite happens. The columns containing the constraints turn yellow and revert to the global values entered in the top of the dialog. The following tabs are available:

- Gravity Pipe
- Gravity Structure

Additional check boxes are available to specify exactly what you want the software to design automatically.

### Design Constraints Alternative Editor for Gravity Pipes

This dialog contains the following sections:

- Minimum and Maximum Default Constraints
  - **Minimum Velocity** - Minimum velocity that is allowed in a pipe.
  - **Minimum Cover** - Minimum depth to which the crown of a pipe must be buried.
  - **Minimum Slope** - Minimum slope of a pipe segment.
  - **Maximum Velocity** - Maximum velocity that is allowed in a pipe.
  - **Maximum Cover** - Maximum depth to which the crown of a pipe may be buried.
  - **Maximum Slope** - Maximum slope of a pipe segment.
- Extended Design
  - **Part Full Design** - Specify what percent capacity you want to design your gravity pipes to. In other words you can specify that you want your pipes to be designed for 50% capacity. Click on the check box adjacent to the **Part Full Design** label and then enter the desired percentage. This value will show up for every pipe in the system. You can then manually adjust specific values in the table on the bottom half of this dialogue.

  - **Allow Multiple Sections** - Specify whether the automatic design should consider the use of multiple parallel pipes, and if so how many. By default the model will only use one pipe section for design.
  - **Limit Section Size** - Specify a maximum section size for design.

- **Table Columns** - The following columns are editable:
  - **Design Pipe** - Specify whether the program should design the pipe based on the constraints given to the model.
  - **Design Upstream Invert** - Specify if the program should design the upstream invert based on the constraints given in the model.
  - **Design Downstream Invert** -Specify if the program should design the downstream invert based on the constraints given in the model.
  - **Specify Local Constraints** - If the box in this column is checked then you can enter local values to replace the default values. If there is not a check mark in the box then the program will automatically use the default constraints.

  The rest of the columns contain the values entered in the **Default Constraints** section, unless the **Specify Local Constraints** box is checked. These are the constraints that govern the automatic design process:

  - **Minimum Velocity**
  - **Maximum Velocity**
  - **Minimum Cover**
  - **Maximum Cover**
  - **Minimum Slope**
  - **Maximum Slope**

  The next six columns are for specifying local values for the Extended Design Options discussed above:

  - **Part Full Design**
  - **Design Percent Full**
  - **Allow Multiple Sections**
  - **Maximum Number Sections**
  - **Limit Section Size**
  - **Maximum Design Section Rise**

For example, to specify a percentage full for a particular pipe, check the appropriate box in the column entitled **Part Full Design**. Then type in the value in the next column entitled **Design Percent Full**.

### Design Constraints Alternative Editor for Gravity Structures

This dialog contains the following sections:

- **Default Design Constraints**
  - **Pipe Matching** - Specify whether the program should match pipe crowns or inverts.
  - **Matchline Offset** - This is used to design invert elevations in and out of a gravity structure. The specified value will produce a corresponding drop between the upstream and downstream

invert elevations. A drop such as 0.1 ft is typically used to compensate for the junction headloss.

  – **Allow Drop Structure** - Permit the program to use drop structures in design mode. If the box next to this label has a check mark then drop structures are permitted; otherwise they are not permitted.

- Table Columns
  – **Structure Type** - Specify if the structure is a junction chamber, manhole, or outlet.
  – **Design Structure Elevation** - Specify whether the sump elevation can be adjusted. If there is a check mark in the cell in this column then you can set the sump elevation of this node, otherwise the program will calculate the sump elevation.
  – **Desired Sump Depth** - Set the sump depth if there is a check mark in the column labeled **Design Structure Elevation**. If there is not a check mark in this column then the program will use the sump depth entered in the respective node dialog.
  – **Local Pipe Matching Constraints** - Assign local values to the design constraints for a node if there is a check mark in the box in this column. If there is not a check mark then the program assigns the default constraints to the node.

The values in the following columns are the same as the default constraints if there is not a check mark in the column labeled **Local Pipe Matching Constraints**. If there is a check mark, you may enter local data.

  – **Pipe Matching**
  – **Matchline Offset**
  – **Allow Drop Structure**.

## 8.2.10 Initial Settings Alternative Editor

The **Initial Settings Alternative Editor** allows you to specify the starting conditions of pressure pipes, pumps, and wet wells. For example, you can specify whether a particular pressure pipe is open or closed. Thereby modeling how the system will react if a certain pipe goes out of service, or if you wish to isolate future portions of the system.

### Initial Settings Alternative Editor for Pressure Pipes

The **Initial Settings Alternative Editor** for pressure pipes is used to specify if pressure pipes are initially open or closed.

### Initial Settings Alternative Editor for Pumps

The **Initial Settings Alternative Editor** for pumps allows you to specify the initial pump settings. The fields for each pump are as follows:

- **Status** - Indicates whether the pump is initially On or Off.
- **Relative Speed Factor** - Specify the initial speed of the pump impeller relative to the speed at which the pump curve is defined.

### Initial Settings Alternative Editor for Wet Wells

The **Initial Settings Alternative Editor** for wet wells allows you to specify the initial wet well settings. The fields for each wet well are as follows:

- **Fixed Level** - Determine how the Hydraulic grade in the wet well is calculated. See the *Hydraulic Transition from Gravity to Pressure Network* topic in Appendix B for more details.
- **Initial Elevation** - The elevation or level of the water surface at the beginning of the simulation.

### 8.2.11 Operational Alternative Editor

The **Operational Alternative Editor** allows you to specify controls on pressure pipes and pumps. The **Controlled** field contains a true or false statement that indicates whether the network element is controlled. Clicking this field activates a button that allows you to edit the controls for the network element.

### 8.2.12 Cost Alternative Editor

One of the most common uses of the **Cost Manager** is to compare the cost between several different system configurations. The compartmentalization of the data afforded by the cost alternative makes it easy to develop and subsequently compare various cost data sets. Developing multiple cost alternatives is an effective way to evaluate the cost of several different proposed solutions or to separate the costs associated with several phases of construction.

The cost alternative editor contains a tab for each type of element. Each tab contains the following fields for editing the cost data associated with an element.

- **Label** - Identifies the element associated with a particular record.
- **Include in Cost Calculation** - This field allows the user to specify whether or not to include the element in the cost calculation. If this field is check the item will be included in the cost calculation.
- **Element Costs** - This field is only enabled when the field **Include in Cost Calculation** has a check mark. If this field is editable you can click on it to open up a dialog where you can edit the construction and non-construction costs associated with an element. Note that you can Global Edit this field to edit the construction and non-construction costs of all the elements in the alternative.

### 8.2.13 User Data Alternative Editor

The **User Data Alternative Editor** allows you to edit the user data you defined in the User Data Extension command for each of the network element types. The **User Data Alternative Editor** contains a tab for each type of network element.

See the section *Hydraulic Element Editors* in the help for more information.

## 8.3 Scenarios

A Scenario contains all the input data (in the form of Alternatives), calculation options, results, and notes associated with a set of calculations. Scenarios let you set up an unlimited number of "*What If?*" situations for your model, and then modify, compute, and review your system under those conditions.

You can create scenarios that reuse or share data in existing alternatives, submit multiple scenarios for calculation in a batch run, switch between scenarios, and compare scenario results - all with a few mouse clicks. There is no limit to the number of scenarios that you can create.

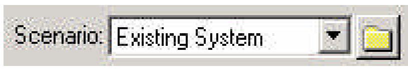
There are two types of scenarios:

- **Base scenarios** - Contain all of your working data. When you start a new project, you will begin with a default base scenario. As you enter data and calculate your model, you are working with this default base scenario and the alternatives it references.

- **Child scenarios** - Inherit data from a base scenario, or other child scenarios. Child scenarios allow you to freely change data for one or more elements in your system. Child scenarios can reflect some or all of the values contained in their parent. This is a very powerful concept, giving you the ability to make changes in a parent scenario that will trickle down through child scenarios, while also giving you the ability to override values for some or all of the elements in child scenarios.

> **The calculation options are not inherited between scenarios, but are duplicated when the scenario is first created. The alternatives and data records, however, are inherited. There is a permanent, dynamic link from a child back to its parent.**

## 8.3.1 Scenario Selection




You can change the current scenario by simply using the **Scenario** drop-down list located on the **Analysis Toolbar** on the main application window. When you select a different scenario, your current input data, calculation options, and calculated results (if available) will reflect the selected scenario and the alternatives it references.

## 8.3.2 Editing Scenarios

Once scenarios and alternatives are created, you do not need to take any special steps to input data into the alternatives referenced by the current scenario. This happens automatically as you make changes to your data. Changes to your data are always applied to the alternatives in your active scenario. For example, consider that a pipe has a 12" diameter in the alternative storing data for the Base scenario. Then you switch to Scenario 2, which references another alternative, and change the pipe diameter to 16". The new value will automatically be associated with the alternative in Scenario 2. If you switch back to the Base scenario, the pipe diameter will revert to 12".

You can also enter data directly into an alternative using the **Alternatives Editor**. This editor allows you to see all of the changes that you have made in a single alternative. If you make an unintended change to the active child scenario and you wish to remove it, go to the tabular editor for the type of input data you changed, and uncheck the leading check box on the record(s) for the elements you wish to restore.

Scenarios currently only track modifications to the input data associated with existing network elements. They do not allow you to track modifications to the network topology itself (e.g. additions and deletions of network elements).

## 8.3.3 Scenario Manager

The **Scenario Manager** allows you to create, edit, and manage scenarios. There is one built-in default scenario - the Base scenario. If you wish, you only have to use this one scenario. However, you can save yourself time by creating additional scenarios that reference the alternatives needed to perform and recall the results of each of your calculations. There is no limit to the number of scenarios that you can create.

The **Scenario Manager** is divided into four sections:

1. The three buttons that run across the top of the window:
   - **Tutorial** - Open the tutorials.
   - **Close** - Close the Scenario Manager.
   - **Help** - Open the on-line help.
2. The series of five buttons running along the left side of the window:

- **Scenario Wizard** - Open the **Scenario Wizard**, which walks you step-by-step through the creation of a new scenario.
- **Scenario Management** - Offers a menu of options for creating, editing, and managing scenarios:

  **Add** - Prompts for a name, then creates a new child or base scenario. If you create a child scenario, it will be based on the scenario that is currently highlighted.

  **Edit** - Open the **Scenario Editor** for the scenario that is currently highlighted.

  **Rename** - Rename an existing scenario. This invokes an in-place editor in the tree view of the available scenarios. Make the desired changes to the existing name and press **Enter**.

  **Delete** - Delete the scenario that is currently highlighted.

  **Report** - Generate a summary report for the scenario that is highlighted, including alternatives, calculation options, notes, and results.

- **Alternatives** - Open the **Alternatives Manager** for creating, editing, and managing alternatives.
- **Batch Run** - Open the **Batch Run** dialog for selecting from among the available scenarios and initiating calculations.
- **Scenario Comparison** - Open the **Annotation Comparison Wizard**, which allows you to create a drawing displaying the differences in input and output variables between two scenarios.

3. The pane in the center of the dialog:
   - **Scenarios Pane** - Available scenarios in a hierarchical tree showing the parent-child relationships. You can right-click any scenario to perform scenario management functions on it. You can double-click parent scenarios to expand or collapse the child scenarios beneath them.
4. The pane on the right side of the dialog, which displays a variety of information depending on which of the following tabs is selected:
   - **Alternatives** tab - Alternatives referenced by the highlighted scenario, showing the type and name for each alternative. An icon distinguishes whether the alternative belongs to the scenario or is inherited from its parent scenario. Double-click any alternative to open the **Alternatives Editor**.
   - **Summary** tab - Summary of the calculation options for the highlighted scenario, and any notes you have associated with it.
   - **Results** tab - Summary of the last calculation performed for the highlighted scenario.

**When you delete a scenario, you are not losing data records because scenarios never actually hold calculation data records (alternatives do). The alternatives and data records referenced by that scenario will still exist until you explicitly delete them. By accessing the Alternative Manager, you can delete the referenced alternatives and data records.**

To open the **Scenario Manager** window, select **Analysis / Scenarios**. Or, click the button next to the scenario drop-down list in the main application window.

## Batch Run

Performing a batch run allows you to set up and run calculations for multiple scenarios at once. This is helpful if you want to queue a large number of calculations, or simply manage a group of smaller calculations as a set. The list of selected scenarios for the batch run will remain with your project until you change it.

Using the dialog is simple. First, check the scenarios you want to run and click the **Batch** button. Each scenario will be calculated. You can cancel the batch run between any scenario calculation.

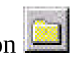
When the batch is completed, the scenario that was current will remain current, even if it was not one that was calculated. Select a calculated scenario from the main window drop-down list to see the results throughout the program, or select it from the **Scenario Manager** and click the **Results** tab to preview the results.

The **Batch Run** dialog contains a **Select** button that displays options you can use to select all scenarios or clear your selections.

### Creating Scenarios to Model "What If?" Situations

The scenario management feature was designed to let you model "W*hat If?*" situations by easily switching between different input data sets without having to re-enter data. Comparing different output results is just as simple.

**To create a new scenario:**

1. Open the **Scenario Manager** dialog by clicking the **Scenario Manager** button  next to the drop-down scenario list in the main application window.
2. Click the **Scenario Wizard** button in the upper left of the **Scenario Manager** dialog.
3. Complete each step in the **Scenario Wizard** - Name the new scenario, choose which scenario to base it on, and choose the alternatives to be included. Click **Next** between each step, and click **Finish** when you are done.
4. Close the **Scenario Manager** dialog. Notice the scenario you have just created is displayed as the current scenario in the **Scenario** choice list in the main application window.
5. Proceed to modify your model with the changes you want recorded in the new scenario.

## 8.3.4 Scenario Wizard

The **Scenario Wizard** will guide you step-by-step through the process of creating a new scenario.

These are the basic steps for creating a new scenario:

- **Name** - Name the scenario and add some comments if you wish.
- **Base** - Select a scenario on which to base the new scenario.
- **Calculation** - Choose the type of calculation that you would like to perform, as well as other calculation options.
- **Alternatives** - Specify the alternative types with which you would like to work.
- **New/Existing** - Create and/or select alternatives for your new scenario.
- **Preview** - Preview the scenario, and create it when satisfied.

To access the **Scenario Wizard**, open the **Scenario Manager** and click the **Scenario Wizard** button.

### Scenario Wizard - Step 1

Here you can enter a unique name and an optional note for the new scenario that you are creating.

The **Name** field allows you to input a distinguishing name for this scenario. A default name is provided, but we recommend that you change it to something more descriptive. If the new scenario will be based on another scenario, you may want a name that indicates what will be different about the new scenario. For example: "Post Development".

The next field is optional, and allows you to input free-form text that will be associated with the new scenario. Use it to make detailed notes about the conditions the scenario will model.

Click the **Next** button to proceed to the next step in defining a new scenario.

### Scenario Wizard - Step 2

Click the existing scenario on which you would like to base your new scenario. Your new child scenario will inherit data from this parent scenario, and will be initialized with the same calculation settings and options. The **Scenario Wizard**, designed to introduce the user to scenarios, does not allow you to create new base scenarios.

Existing scenarios in your project are displayed in a "tree" structure, giving you a graphic depiction of the parent-child relationships.

Click the **Next** button to proceed to the next step.

### Scenario Wizard - Step 3

This tab of the **Scenario Wizard** allows you to specify the type of calculation to be associated with the scenario you are creating, either Extended Period or Steady State.

You can choose to perform a Design for a Steady State run, and also specify the Extreme Flow Setup and the Pattern Setup.

### Scenario Wizard - Step 4

Check the boxes next to the types of alternatives you want to include in the new scenario. The alternatives for boxes you do not check will be inherited from the specified parent scenario. You will be free to add or remove alternatives to the scenario after you create it.

Click the **Next** button to proceed to the next step in defining a new scenario.

### Scenario Wizard - Step 5

Here you are asked to specify the source for each alternative you have requested in the previous tab.

- **Create New Alternative -** If you choose to create a new alternative, it will inherit from the same type of alternative in the specified or parent scenario. This means it will initially use all the same input data values. Enter a unique and descriptive name for the new alternative.
- **Use Existing Alternative -** If you choose to use an existing alternative, you will be shown the tree of existing alternatives from which to choose. In this case, you will not be creating a new alternative for use in the scenario, and instead may actually be *sharing* an alternative with another scenario.

Click the **Next** button to proceed to the next step.

### Scenario Wizard - Step 6

The last step of the **Scenario Wizard** displays a summary of the scenario you have defined and are about to create.

In the left pane is a preview of the scenario as it relates to its parent and other scenarios. In the right pane is a list of the alternatives it references, showing their labels and types. An icon indicates whether a given alternative is local to the new child scenario, or if it is inherited from the specified or parent scenario.

If you are satisfied, click the **Finished** button to create the new scenario.

## 8.3.5 Scenario Editor

The **Scenario Editor** is the control center for each analysis. It is the place where you access or change all the information for performing a single calculation (alternatives, calculation type, calculation options, results, and notes). It is organized by the following tabs:

- **Alternatives** - Edit or view the alternatives to be used by this scenario.
- **Calculation** - Specify the type of hydraulic/water quality calculations to be performed, and click the **GO** button.
- **Results** - View the hydraulic/water quality calculation results summary.
- **Notes** - Edit or view notes for this scenario.

To open the **Scenario Editor** for the active scenario, click the **GO** button on the toolbar. To open the **Scenario Editor** for any scenario, select **Analysis\Scenarios** from the pull-down menu to open the **Scenario Manager**, right-click the scenario that you wish to edit, and select **Edit** from the pop-up menu. Or, highlight the scenario you wish to edit, click the **Scenario Management** button, and select **Edit**.

### Alternatives Tab

The **Alternatives** tab, located in the **Scenario Editor**, allows you to specify the alternatives that will be used by this scenario. There is one row for each Alternative Type. You need only concern yourself with the rows that correspond to the changes you would like to model using this scenario.

To specify the alternatives you would like to work with, simply click the check box next to the alternative type. For example, if you would like to see how your system behaves by changing the shape or size of a few pipes, then click the check box next to the Physical Properties Alternative row.

If you would like to use an existing alternative that you have already set up, use the drop-down list to choose the desired alternative. If you would like to create a new alternative, click the **New** button. You will be asked to name the new alternative, and the **Alternatives Editor** will open.

The **Scenario Wizard** will walk you through all of the steps required to create a new scenario. If you are unsure how to specify the alternatives that you would like to work with, we recommend that you use this wizard.

**When this scenario is active, the alternatives that you specify here will be active. Changes that you make to your model will be made in these alternatives. When you calculate this scenario, these are the alternatives that will be used.**

**This tab will take on a different appearance depending on whether you are editing a base scenario or child scenario. When editing a base scenario, the checkbox column described above will not be present. You can use the ellipsis (…) button located to the right of each drop-down list to access the associated Alternatives Manager.**

### Calculation Tab

The **Calculation** tab, located on the **Scenario Editor**, lets you define the type of calculation and analysis you wish to perform for this scenario, as well as setting various calculation options.

To calculate the model you must perform the following steps:

1. Choose from either Steady State or Extended Period.
2. If performing a Steady State analysis select the following parameters:
   - Check **Design** if you wish SewerCAD to perform an Automated Design on the gravity portion of the network.

- Select an Extreme Flow Setup from the **Extreme Flow Setup Manager**.

3. If performing an Extended Period analysis select and enter the following parameters:
   - **Duration** - Specify the duration of the Extended Period Simulation.
   - **Hydraulic Time Step** - Specify the time increment between hydraulic backwater analyses in the gravity portions of the network.
   - **Hydrologic Time Step** - Specify the routing increment used to generate hydrographs in the gravity system, and the time increment used in the pressure portions of the network.
   - Select a Pattern Setup from the **Pattern Setup Manager.**

4. Click the **GO** button.

The Physical Properties and Design Constraints alternatives used for the design will be displayed.

**In Stand-Alone mode, if the model has not been calculated, or if the input data has been changed since the last calculation, the word COMPUTE (highlighted in Red) will appear in the status pane in the lower right corner of the main application window. This is a signal that the model needs to be recalculated.**

## Results Tab

The **Results** tab contains a summary of the last calculation performed using this scenario. Click the **Save** button to save the results to an ASCII text file. Click the **Print Preview** button to preview the Scenario Results Summary Report.

To open the **Results** tab for the active scenario: Click the **GO** button on the toolbar, or select **Analysis \ Compute** from the pull-down menu. In the resulting dialog, click the **Results** tab.

To open the **Results** tab for a specific scenario: From the **Scenario Manager**, right-click the scenario that you wish to edit, and select **Edit** from the pull down menu that appears. In the resulting dialog, click the **Results** tab.

**Immediately after you run the calculations, the Results tab displays automatically. You will notice a green, yellow, or red light in that tab indicating how successful the computations were.**

The light and folder color provides you with the following information:

**Green light** - Calculations were run successfully, without any warning or error messages being generated.

**Yellow light** - Calculations were run successfully, without error messages being generated. However, there are one or more warning messages. Warnings are displayed in the results summary in this tab.

**Red light** - Calculations were not run successfully and error messages were generated, as shown in the results summary of this tab.

In order to generate a Scenario Summary Report, click the **Report** button. Click the **Element Messages** button to open up a table that displays all the messages generated during the run.

Double click the folders or click the **+** sign to open them up and display messages relevant to the folder's caption. When you click the copy button or save button the exposed text will be stored.

## Notes Tab

The memo field on the **Notes** tab allows you to input paragraph text that will be associated with the new scenario. Use it to make detailed notes about the conditions that the scenario will model.

Notes

# Chapter 9

# Modeling and Design Capabilities

## 9.1 Calculate

To run network calculations, you must set parameters in the following sections on the **Calculation** tab of the **Scenario Editor**:

- **Calculation Type** - Perform either a Steady State or Extended Period analysis.
- **Steady State** - Choose whether or not to run a design and select an Extreme Flow Setup from the **Extreme Flow Setup Manager.** Clicking the **ellipsis (…)** button in the **Extreme Flow Setup** field will open the **Extreme Flow Setup Manager**.
- **Extended Period -** Enter a Duration, Hydraulic Time Step, and a Hydrologic Time Step. Also, select a Pattern Setup from the **Pattern Setup Manager**. Clicking the **ellipsis (…)** button in the **Pattern Setup** field will open the **Pattern Setup Manager**.

Click the **Options** button to check or change calculation settings. After exiting the **Calculation Options** dialog, click the GO button to start the calculations.

**The Check Data button performs a quick check of your input data and displays any errors found. This function is automatically performed when you run a calculation.**

### 9.1.1 Calculation Type Section

This section lets you specify whether you are in Steady State or Extended Period mode.

- **Steady State** - Runs the model for a single instant in time with extreme flow factors applied.
- **Extended Period** - Runs the analysis over a specified duration of time with loading patterns applied.

### 9.1.2 Steady State Section

This section allows you specify whether you would like to perform an automated design and lets you select an Extreme Flow Setup.

If **Design** is checked, all pipe invert elevations, diameters, and structures that are selected for design will have the corresponding values updated automatically.

**The values that may change due to the design process are stored in the Physical Alternative, using the Design Constraints specified in the Design Constraints alternative. When you are ready to do the calculation, you will be asked if you want to save this new data in a separate Physical Alternative, in case you want to preserve your current Physical data.**

In the **Extreme Flow Setup** field you can specify which previously created Extreme Flow Setup to associate with the Steady State run. Clicking the **ellipses (…)** button will open the **Extreme Flow Setup Manager**.

### 9.1.3 Extended Period Section

This section lets you specify the following parameters necessary to run the Extended Period Simulation.

- **Duration** - Length of simulation.
- **Hydraulic Time Step** - Time increment between gravity backwater analyses.
- **Hydrologic Time Step** - Time increment between computed hydrograph ordinates. This also serves as the increment used in the pressure calculations.
- **Pattern Setup** - In this field you can specify which previously created Pattern Setup to associate with the Extended Period run. Clicking the **ellipses (…)** button will open the **Pattern Setup Manager**.

## 9.2 Calculations Options

Calculations depend upon a variety of parameters that may be configured by you.

This program provides defaults for each of the calculation options. If you make changes to the calculation options and decide that you would like to return to the default settings, use the **Reset** button on the **Calculation Options** dialog.

The dialog is divided into seven tabs:

- Gravity Hydraulics
- HEC-22
- AASHTO
- Generic Structure Loss
- Convex Routing
- Steady State Loading
- Pressure Hydraulics

To access the **Calculation Options** dialog, click the **Options** button in the **Scenario Editor** , or select the 

button and click the **Options...** button.

### 9.2.1 Gravity Hydraulics Tab

The **Gravity Hydraulics** tab is the place where you can adjust the hydraulic options relating to the gravity portion of the calculation. The following sections are available on the **Gravity Hydraulics** tab:

- Flow Profile Method
- Hydraulic Grade Convergence Test
- Average Velocity Method
- Minimum Structure Headloss

### Flow Profile Method Section

This section lets you specify whether you are performing a Backwater Analysis or a Capacity Analysis for the gravity portions of the network.

If you select the backwater analysis, you need to specify the **Number of Flow Profile Steps**. The gradually varied flow profile within each pipe divides the pipe into internal segments prior to calculation of the hydraulic grade. The default value is five, and it is recommended that the value entered here be at least five segments for accuracy. Increasing this number will increase the accuracy of the hydraulic grade calculation, but will also make the network take longer to calculate.

**The higher the number of steps, the more accurate the results, but the slower the calculations. A value of five steps should be accurate enough in most cases.**

### Hydraulic Grade Convergence Test Section

This section contains the tolerance value constraining the Hydraulic Grade Convergence Test. In a full network calculation, this value is taken as the maximum absolute change between two successive solutions of the hydraulic grade at any junction or manhole in the system. This test is used to optimize the performance of the system solutions. It minimizes the number and extent of hydraulic grade line computations in the upstream direction. For a given discharge, the upstream propagation of headlosses through pipes will continue until two successive calculations change by an absolute difference of less than this test value.

HGL Convergence Test value is also used in the standard step gradually varied flow profiling algorithm. If two successive depth iterations are within this absolute test value, the step is solved.

### Average Velocity Method Section

This section allows you to choose the method used to calculate the velocity and the travel time through the pipe. You have the following four options:

- Actual Uniform Flow Velocity
- Full Flow Velocity
- Simple Average Velocity
- Weighted Average Velocity

**You can find more information on the different velocity methods in Appendix B of the help.**

### Minimum Structure Headloss

This section allows you to specify a minimum structure headloss. If the system calculates a structure headloss that is lower then this value, the value specified in the **Minimum Headloss** field will be used. This option applies to all structure headloss methods except for the Absolute Method. Absolute headlosses will not be overridden, even if they are less then the value specified in this option.

## 9.2.2 HEC-22 Tab

This tab is where you enter the values governing the calculations of the HEC-22 Energy Loss Method. The specifics of this tab are presented below:

- **HEC-22 Energy Loss Method** - Enter the **Elevations Considered Equal Within** value, which is the maximum elevation distance that pipes entering a node can be separated by and still be considered to be at the same elevation.

- **Correction for Benching** - Contains values for the items in the following list governing the correction for benching sections:
  - Flat Submerged
  - Flat Unsubmerged
  - Depressed Submerged
  - Depressed Unsubmerged
  - Half Bench Submerged
  - Half Bench Unsubmerged
  - Full Bench Submerged
  - Full Bench Unsubmerged

**You can find more information on the HEC-22 Energy Loss Method in Appendix B of the help.**

## 9.2.3 AASHTO Tab

This tab is where you can enter the values governing the calculations of the AASHTO Headloss Method. The following sections are available:

- Bend Angle and Loss section
- Other Factors section

**You can find more information on the AASHTO Headloss Method in Appendix B of the help.**

### AASHTO Bend Angle and Loss Section

This section allows you to enter the bend angles and associated bend loss coefficients, Kb, that are used in the calculation of headloss in the AASHTO Headloss Method.

### AASHTO Other Factors Section

In this section, you can enter the values for the other factors that govern the calculation of headlosses using the AASHTO Headloss Method. These factors are presented in the following list:

- Expansion
- Contraction
- Shaping Adjustment
- Non-Piped Flow Adjustment

## 9.2.4 Generic Structure Loss Tab

### Calculation Options Generic Structure Loss Tab

On this tab, you can change the methodology for selecting the upstream pipe when computing the headloss for a structure using the Generic Headloss Method. The three methodologies are described below.

- **Pipe With Maximum QV** - If this item is selected, the program will use the non-plunging upstream pipe with the largest flow times velocity to calculate the upstream velocity head used in the generic headloss equation.
- **Pipe With Minimum Bend Angle** - If this item is selected, the program will use the upstream pipe with the smallest bend angle to calculate the upstream velocity head used in the generic headloss equation. The methodology should be used when you want to assume that the upstream pipe most closely aligned with the downstream pipe is the one that is the most hydraulically significant.
- **Pipe With Maximum Velocity Head** - If this item is selected, the program will use the non-plunging upstream pipe with the largest velocity head to calculate the upstream velocity head used in the generic headloss equation. Note that if this method is used, pipes with very small flows may be selected as the governing pipe, even though they are not hydraulically significant.

The methodology that is selected here will be used for all structures that employ the generic headloss method.

## 9.2.5 Convex Routing Tab

In this section you can adjust the **Peak Flow Ratio** for use when selecting a representative flow rate from the hydrograph to be routed when calculating the C parameter used to perform the routing calculations.

**You can find more information on the routing methods used in SewerCAD in Appendix B of the help.**

## 9.2.6 Steady State Loading Options Tab

In this section the user specifies how SewerCAD should handle hydrographs during a steady state analysis. One of the following methods can be selected:

- **Peak** - The peak of the hydrograph will be used as the load.
- **Average** - The average of all the flows in the hydrograph will be used as the load.
- **Minimum** - The lowest flow of the hydrograph will be used as the load.
- **Zero** - The hydrographs will be ignored during the analysis.

## 9.2.7 Pressure Hydraulics Tab

The **Pressure Hydraulics** tab is the place where you can adjust the hydraulic options relating to the pressure portion of the calculation. For more information on any of the options on this tab, see the following sections:

- Pressure Options
- Gravity Pressure Interface Options

### Pressure Options Section

Specify the hydraulic settings controlling Pressure Flow computations, as follows:

- **Trials** - A unitless number that defines the maximum number of iterations to be performed for each hydraulic solution. The default value is 40.
- **Accuracy** - A unitless number that defines the convergence criteria for the iterative solution of the network hydraulic equations. When the sum of the absolute flow changes between successive iterations in all links is divided by the sum of the absolute flows in all links, and the result is less than the Accuracy, the solution is said to have converged. The default value is 0.001, and the minimum allowed value for Accuracy is $1.0 \times 10^{-5}$.

- **Use Controls in Steady State Analysis** - When this toggle is on, the pressure subnetwork will be evaluated after each trial to determine if any controls are triggered by changes to the system hydraulics. This option enables pumps to change status or setting, and enables pipes to open or close. If this option is not selected, controls will be ignored. Pump and pipe status will be based on initial status and settings only.

**In most cases, the default values are adequate for the hydraulic analysis. Under special circumstances, the accuracy may need to be adjusted downwards. This is necessary when the model converges yet there are larger than acceptable discrepancies between the total inflow and total outflow at individual nodes.**

### Gravity Pressure Interface Options Section

In this section, you can specify the parameters affecting the transition from a gravity network to a pressure system during a Steady State analysis, as follows:

- **Wet Well Increment** - Unless a wet well is set to a Fixed Level, this is the increment that is used to attempt to balance the wet well level such that the total flow out is equal to or greater than the total flow in. See the Hydraulic Transition from Gravity to Pressure Network in Appendix B for a further explanation.
- **Use Pumped Loads** - Loads discharging from a pressure subnetwork into a gravity subnetwork can be treated as fixed flow loads by selecting the option **Use Pumped Flows**. If this option is not selected, the discharging load will be recreated as a combination of dry, wet, and known loads (in accordance with the loads entering the pressure subnetwork).

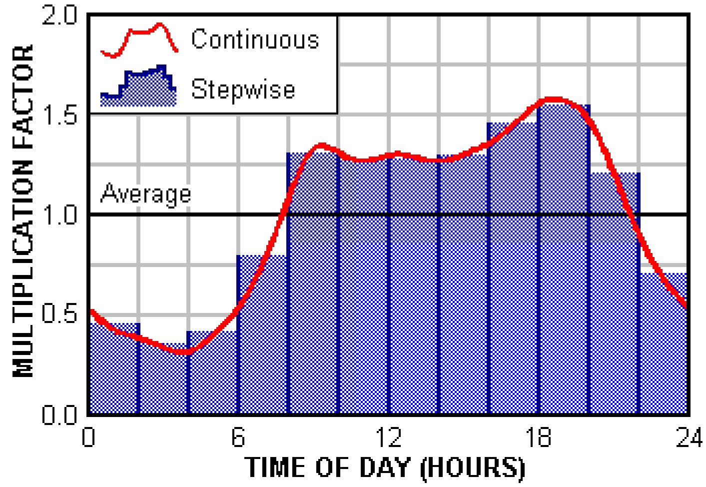
# 9.3 Pattern Manager

## 9.3.1 Patterns

The extended period analysis is actually a series of Steady State analyses run against time-variable loads such as sewer inflows, demands, or chemical constituents. Patterns allow you to apply automatic time-variable changes within the system. The most common application of patterns is for residential or industrial loads. Diurnal curves are patterns that relate to the changes in loads over the course of the day, reflecting times when people are using more or less water than average. Most patterns are based on a multiplication factor versus time relationship, whereby a multiplication factor of one represents the base value (which is often the average value).

Using a representative diurnal curve for a residence as illustrated below, we see that there is a peak in the diurnal curve in the morning as people take showers and prepare breakfast, another slight peak around noon, and a third peak in the evening as people arrive home from work and prepare dinner. Throughout the night, the pattern reflects the relative inactivity of the system, with very low flows compared to the average.

Typical Diurnal Curve




> **This curve is conceptual and should not be construed as representative of any particular network.**

There are two basic forms for representing a pattern: stepwise and continuous. A stepwise pattern is one that assumes a constant level of usage over a period of time, and then jumps instantaneously to another level where it remains steady until the next jump. A continuous pattern is one for which several points in the pattern are known and sections in between are transitional, resulting in a smoother pattern. For the continuous pattern in the figure above, the multiplication factor and slope at the start time and end times are the same. This is a continuity that is recommended for patterns that repeat.

Because of the finite time steps used for calculations, this software converts continuous patterns into stepwise patterns for use by the algorithms. In other words for a time step a multiplier is interpolated from the pattern curve. That multiplier is then used for the duration of the time step, until a new multiplier is selected for the next time step.

Patterns provide a convenient way to define the time variable aspects of system loads.

## 9.3.2 Pattern Manager

Patterns provide an effective means of applying time-variable system loads to the distribution model. The **Pattern Manager** allows you to do the following:

- **Add** - Click the **Add** button. This action opens the **Pattern Editor** where the specifics of the pattern can be entered.
- **Edit** - Select the label of the pattern you wish to edit, and click the **Edit** button. The **Fixed** pattern cannot be edited.
- **Duplicate** - Select the label of the pattern you wish to duplicate, and press the **Duplicate** button. The **Fixed** pattern cannot be duplicated.
- **Delete** - Select the label of the pattern you wish to delete, and click the **Delete** button. The **Fixed** pattern cannot be deleted.

> **In this program, an individual loading node can support multiple hydraulic loads. Furthermore, each load can be assigned any hydraulic pattern. This powerful functionality makes it possible to model any type of extended period simulation.**

To access the **Pattern Manager** select **Analysis / Patterns** from the pull-down menus, or click the **ellipsis (…)** button next to any pattern choice list.

### 9.3.3 Pattern Editor

A pattern is a series of time step values, each having an associated multiplier value. During an extended period analysis each time step of the simulation uses the multiplier from the pattern corresponding to that time. If the duration of the simulation is longer than the pattern, the pattern is repeated. The selected multiplier is applied to any baseline load that is associated with the pattern.

**Defining Patterns**

- **Label** - A required name to uniquely identify the pattern. This name appears in the choice list when applying patterns to hydraulic demands or constituent source loads.
- **Start Time** - A value between 0 and 24 that specifies the first time step point in the pattern. All other pattern time step points are referenced from this start time. This program automatically adjusts your pattern when you start an Extended Period Analysis at a time other than zero.
- **Starting Multiplier** - The multiplier value of the first time step point in your pattern. Any real number can be used for this multiplier (it does not have to be 1.0).

**Time Step Points**

- **Time From Start** - The amount of time from the **Start Time** of the pattern to the time step point being defined.
- **Multiplier** - The multiplier value associated with the time step point.

**Format**

- **Stepwise** - The multiplier values are considered to be the average value for the interval between the specified time and the next time. Patterns using this format will have a "staircase" appearance. Multipliers are set at the specified time and held constant until the next point in the pattern.
- **Continuous** - The multipliers are considered to be the instantaneous values at a particular time. Patterns using this format will have a "curvilinear" appearance. Multipliers are set at the specified time, and are linearly increased or decreased to the next point in the pattern.

**Patterns must begin and end with the same multiplier value. This is because patterns will be repeated if the duration of the Extended Period Analysis is longer than the pattern duration. In other words, the last point in the pattern is really the start point of the pattern's next cycle.**

**An Extended Period Analysis is actually a series of Steady State analyses for which the boundary conditions of the current time step are calculated from the conditions at the previous time step. This software will automatically convert a continuous pattern format to a stepwise format so that the demands and source concentrations remain constant during a time step.**

**An individual loading node can support multiple hydraulic loads. Furthermore, each load can be assigned any hydraulic demand pattern. This powerful functionality makes it easy to combine two or more types of demand patterns (such as residential and institutional) at a single loading node.**

**Use the Report button to view or print a graph or detailed report of your pattern.**

To access the **Pattern Editor**, click the **Add** or **Edit** button on the **Pattern Manager**.

### 9.3.4 Pattern Graph and Report

You can generate a graph or a full report of a pattern that represents the multiplier variable of the pattern over time. To do so, open the appropriate **Pattern Manager** dialog and access the pattern for which you would like to generate output. From the Pattern Editor dialog, click the **Report** button, and select **Graph** or **Detailed Report**.

## 9.4 Pattern Setup Manager

The **Pattern Setup Manager** lets you define a list of Pattern Setups. A Pattern Setup allows you to match unit sanitary (dry weather) loads with appropriate loading patterns. A Pattern Setup is associated with each scenario as specified in the **Calculation** tab of the **Calculate** dialog. Each scenario can use a different Pattern Setup, thus allowing you to model different loading alternatives for different extended period simulations.

- **Add** - Click the **Add** button. This will open the **Pattern Setup** dialog where the specifics of the pattern can be edited.
- **Edit** - Select the label of the pattern setup you wish to edit, and click the **Edit** button.
- **Duplicate** - Select the label of the pattern setup you wish to duplicate and click the **Duplicate** button.
- **Delete** - Select the label of the pattern setup you wish to delete, and click the **Delete** button.

The **Pattern Setup Manager** can be accessed by selecting **Analysis / Pattern Setups** from the pull-down menus, or from the **Calculation** tab of the **Calculate** dialog.

### 9.4.1 Pattern Setup Editor

The **Pattern Setup** can be named by editing the **Label** field.

The **Pattern Setup Editor** contains two tabs:

- **Used Loads** - Contains the unit sanitary (dry weather) loads that are currently being used in the project.
- **Unused Loads** - Contains the loads not currently associated with any node accepting loads (manholes, wet wells and pressure junctions) in the current project. This dialog also lets you select the loading pattern associated with each currently unused load.

> **If you have not yet associated any loads with manholes, wet wells, or pressure junctions, the Used Loads tab list will be empty. In that case, you can either select the loading patterns in the Unused Loads tab, or associate loads to specific nodes first. Then come back to the Used Loads tab to select the diurnal patterns associated with the currently used loads.**

The following columns are contained in the table on each of the tabs:

- **Unit Load** - The name of the unit sanitary load to be associated with a **Pattern Setup**.
- **Diurnal Pattern** - A list of the currently defined patterns in the **Pattern Managers**. You can access the **Pattern Manager** by clicking the **ellipsis (…)** button.

> **If no diurnal patterns are selected, the program will assume that the loads are fixed over the entire time period.**

## 9.5 Extreme Flow Setup Manager

The **Extreme Flow Setup Manager** lets you define a list of Extreme Flow Setups. An Extreme Flow Setup allows you to match unit sanitary loads with appropriate extreme flow factor methods. The Extreme Flow Setup is associated with each scenario as specified in the **Calculation** tab of the **Calculate** dialog. Each scenario can use a different Extreme Flow Setup, thus allowing you to model different load alternatives (average, minimum daily, maximum daily, etc).

- **Add** - Click the **Add** button. This will open the **Extreme Flow Setup** dialog where the specifics of the control can be edited.
- **Edit** - Select the description of the Extreme Flow Setup you wish to edit, and click the **Edit** button.
- **Duplicate** - Select the description of the Extreme Flow Setup you wish to duplicate and click the **Duplicate** button.
- **Delete** - Select the description of the Extreme Flow Setup you wish to delete, and click the **Delete** button.

The **Extreme Flow Setup Manager** can be accessed by selecting **Analysis / Extreme Flows** from the pull-down menu, or from the **Calculation** tab of the **Calculate** dialog.

### 9.5.1 Extreme Flow Setup Editor

The **Extreme Flow Setup Editor** contains two tabs as follows:

- **Used Loads** - Contains the unit sanitary loads that are currently being used in the project.
- **Unused Loads** - Contains the loads not associated with any node accepting loads (manholes, wet wells and pressure junctions) in the current project. This dialog also lets you select the extreme flow factor method associated with each currently used and unused load.

**If you have not yet associated any loads with manholes, wet wells, or pressure junctions, the Used Loads tab list will be empty. In that case, you can either select the extreme flow methods in the Unused Loads tab, or associate loads to specific nodes first. Then come back to the Used Loads tab to select the extreme flow methods associated with the currently used load.**

The following columns are contained in the tables on both tabs:

- **Unit Load** - The sanitary load to be associated with an **Extreme Flow Method**.
- **Loading Unit Type** - Reports the type of loading corresponding to each unit load, as defined in the **Unit Sanitary (Dry Weather) Load Library**.
- **Extreme Flow Method** - Lists the currently defined methods in the **Extreme Flow Factor Method Library**. You can edit this library by clicking the **ellipsis (…)** button.
- **Constant** - Available only for the **Constant Extreme Flow Method**, it simply multiplies the unit load by that constant.
- **Adjustment Multiplier** - Used to adjust an existing **Extreme Flow Method**. The following formula is used:

$$EFF = (EFF - 1)/AM + 1$$

Where EFF is the Extreme Flow Factor used to transform the base (average) load to an extreme load.

The **Adjustment Multiplier** does not change the **Extreme Flow Method**, but changes the extreme flow factor for the selected unit sanitary load. **None** and **Constant Extreme Flow Methods** cannot be adjusted.

**If no extreme flow factor method is selected, the program will assume that the loads are not transformed. In this case, a warning will be issued when you attempt to calculate the system.**

# 9.6 Default Design Constraints

Pipes diameters, invert elevations, and gravity structures can all be designed with the same set of design constraints. You also have the option to adjust these values individually for each pipe or structure.

The **Default Design Constraints** dialog box is divided into two tabs:

- Gravity Pipe
- Gravity Structure

## 9.6.1 Gravity Pipe Tab

The **Gravity Pipe** tab allows you to enter default constraints to be used for the design of pipes when performing a calculation run in design mode. The following sections are available:

- Default Constraints
- Extended Design

## 9.6.2 Default Constraints Section

In this section, you can specify the following default constraints to be used for the design of gravity pipes:

- Minimum and Maximum Velocity
- Minimum and Maximum Cover
- Minimum and Maximum Slope.

The **Default Design Constraints** dialog can be accessed by selecting **Analysis / Default Design Constraints** from the pull-down menu.

## 9.6.3 Extended Design Section

This section lets you specify if the following design parameters are to be used. If they are to be used, you can also specify the associated default value:

- **Part Full Design** - Allows you to specify the Design Percent Full target to be used by the design algorithm.
- **Allow Multiple Sections** - Allows the design algorithm to use more than one identical section in parallel, up to the specified Maximum Number Sections.
- **Limit Section Size** - Limits the pipe section height to a Maximum Design Section Rise value during the design process.

### 9.6.4 Gravity Structure Tab

This tab lets you specify the default design constraints for all gravity structures when performing calculations in design mode. During an automatic design, the program will adjust the elevations of the pipes adjacent to the structure according to the structure's matching constraints. The two choices for matching are Inverts and Crowns. Additionally, the downstream pipe can be offset from the upstream pipe(s) by a specified amount. This value is called the Matchline Offset. Optionally, the program supports the design of drop structures. In some situations, drop structures can minimize pipe cover depths while maintaining adequate hydraulic performance.

# Chapter 10

# Cost Estimating

## 10.1 Overview

The **Cost Manager** allows you to calculate a planning level estimate of the capital costs associated with an entire system or any portion of a system. This makes it easy to compare the costs associated with various scenarios, thus helping to ensure that the most cost-effective design is chosen.

The costs associated with a particular element are broken down into two categories: construction costs and non-construction costs. The total cost for each element is simply the sum of the total construction and non-construction costs. The total cost for a scenario is computed by summing the total cost for every element selected to be included in the cost analysis, and then applying any global cost adjustments that you have defined.

Each construction cost item is expressed as a combination of a quantity, unit, and unit cost. The total cost associated with a single construction cost item is the quantity multiplied by the unit cost. The unit cost for each construction cost item can either be entered directly by you, or if the element is a pipe or gravity structure (e.g. inlet, manhole, junction, junction chamber) it can be calculated based on a Unit Cost Function. A Unit Cost Function is a way to relate a property of the element, such as the diameter of a pipe, to the unit cost. This makes it easy to assign a Unit Cost Function to an element. The cost of the element is then automatically updated when you modify the physical characteristics of the system.

The other type of cost is non-construction. Non-construction cost items are specified as either a lump sum or as a percentage of the total construction costs. This type of cost can be useful when trying to explicitly account for itmes like omissions and contingencies.

In addition to specifying the costs for each element in the system, you can also make adjustments on a system level to the total cost of all the elements included in the cost analysis. This makes it easy to account automatically for contingencies and adjustments on a scenario level.

**You do not need to have a hydraulically valid network to perform a cost analysis. You can quickly calculate the cost associated with a system at any time through the Cost Manager.**

## 10.2 Cost Manager

The **Cost Manager** allows you to quickly compute and compare the costs associated with your different scenarios. This dialog provides you with a convenient place to view, edit, and calculate project level cost data. This dialog is divided into three sections that are described below:

- **Button Section** - This column of buttons provides access to the key pieces of data involved in a cost analysis.
- **Center Pane** - This pane displays an explorer view of the cost information for various scenarios.

– **Left Pane** - This pane displays the contents of the item selected in the center pane.

To open the **Cost Manager**, select **Analysis / Compute Costs** from the pull-down menu, or click the button on the **Analysis** toolbar.

## 10.2.1 Cost Manager - Button Section

On the right side of the **Cost Manager** is a column of buttons that provide access to the key pieces of data involved in a cost analysis. Each of these buttons is described as follows:

- **Unit Cost Functions** - Opens the **Unit Cost Function Manager,** which is the place to add new or edit existing functions describing the relationship between a model attribute and the unit price for the element. For pipes, this might be a table of data relating the pipe material to the cost per unit length.
- **Cost Alternatives** - Opens the **Capital Cost Alternatives Manager** where you can quickly create different cost alternatives. For example, you may wish to compare the cost associated with different cost functions, or to separately calculate the cost of different phases of construction.
- **Cost Adjustments** - Opens the **System Cost Adjustments Table** for the selected scenario. This is the location where you enter adjustments that you wish to make on a scenario level.
- **Active Scenarios** - Opens the **Active Cost Scenarios** dialog where you can select which scenarios will appear in the **Cost Manager**.
- **Cost Reports** - Opens a menu that provides access to one of the predefined cost reports detailing the costs associated with a particular scenario. The reports that can be opened through this button include: detailed report , element summary report, project summary report, pipe costs report, and warnings report.

To open the **Cost Manager**, select **Analysis / Compute Costs** from the pull-down menu, or click the button on the **Analysis** toolbar.

## 10.2.2 Cost Manager - Center Pane

When you open the **Cost Manager**, this pane will contain an explorer view of all the scenarios in the file. The total cost of each scenario is displayed to the right of the scenario label. If cost data was specified for a scenario, you will see a small '+' symbol to the left of the folder icon. You can click this symbol to get an expanded view of the costs associated with a scenario.

In the first level of the expanded view you will see the subtotal for each type of element included in the cost analysis, as well as the total cost adjustments made to the scenario. If you expand any of these items you will get a view of the costs of each individual element. If you expand the view one more level you will be able to see the construction and non-construction costs associated with an element. The contents of any component that is selected will be displayed in the table on the pane to the left of this one.

Just above the right side of this pane is a row of three buttons, which access the following functionality:

- **Properties** - Opens the System Cost Adjustments Table for the currently selected scenario.
- **Graph** - Opens a pie chart of the items comprising the total cost of the scenario.
- **Report** - Opens a tabular report on any component selected in this pane.

To open the **Cost Manager**, select **Analysis / Compute Costs** from the pull-down menus, or click the button on the **Analysis** toolbar.

### 10.2.3 Cost Manager - Left Pane

This pane on the left side of the **Cost Manager** is used to display an expanded view of the contents of the item selected in the center pane.

To open the **Cost Manager**, select **Analysis / Compute Costs** from the pull-down menus, or click the  button on the **Analysis** toolbar.

### 10.2.4 System Cost Adjustments Table

The **System Cost Adjustments Table** allows you to make adjustments to the total cost calculated for all the elements included in the cost analysis. This may include items such as omissions and contingencies that might be represented as a percentage of the total construction costs, or land acquisition costs that are represented as a lump sum. Each cost adjustment consists of the following items:

- **Label** - A unique name that identifies each cost adjustment.
- **Operation** - The mathematical operation that should be used with the factor to compute the cost adjustment.
- **Factor** - A numeric value that is used with the operation and the total scenario cost to compute the cost adjustment.

The types of operations that are supported are described below:

- **% of Construction** - Computes the cost of the adjustment as a percentage of the total construction costs for the scenario. For example, if the total construction cost for a scenario is $100,000 and the numeric value in the factor field is 10, the cost adjustment is $10,000.
- **% of Total Cost** - Computes the cost of the adjustment as a percentage of the total construction and non-construction costs for a scenario. For example, if the total cost for all the elements in a scenario is $200,000 and the numeric value of the factor is 15, the cost adjustment is $30,000.
- **Add** - Adds the numeric value specified for the factor to the other costs computed for a scenario.
- **Lump Sum** - The numeric value specified is a lump sum that is added to the other costs for the scenario.
- **Multiply** - Multiplies the numeric value in the factor field by the total construction and non-construction costs for a scenario.
- **Subtract** - Subtracts the numeric value in the factor field from the other costs computed for the scenario.

To open the **System Cost Adjustments Table**, select **Analysis / Compute Costs** from the pull-down menu, or click the  button on the **Analysis** toolbar. In the **Cost Manager**, click the button labeled **Cost Adjustments**.

## 10.3 Unit Cost Functions

### 10.3.1 Unit Cost Functions Manager

You can add, delete, and edit the Unit Cost Functions for your project through this manager. You will be able to assign the cost functions defined here to one or more of the elements of the appropriate type in your system. For example, if you define a cost function for pipes, you will be able to select this cost function from the choice list on the **Cost** tab of the **Pipe Element Editor**.

Use the **Save** command to save the Unit Cost Functions listed in the **Unit Cost Functions Manager**. You can then import them into another project using the **Import** command. The **Save** and **Import** commands are accessed from the **File** button in this dialog.

## 10.3.2 Tabular Unit Cost Function

This tab contains the data for Unit Cost Functions defined with tabular data. The information is defined in the following fields:

- **General** - Contains general information identifying the Unit Cost Function.
- **Attribute Value Range** - Displays the range of the selected attribute in the current scenario. This information can be useful for specifying cost data for the entire range of values in the model.
- **Unit Cost Data** - Specifies the tabular data relating unit cost to the value of the selected attribute.

In order to help you enter and visualize the function, use one of the following buttons at the bottom of the dialog:

- **Plot** - Plots the tabular data relating cost to the value of the selected attribute.
- **Initialize Range** - Initializes the minimum and maximum values in the **Attribute Value Range** section, based on all the elements present in your project for the current scenario.

**If the attribute you have selected to define the Unit Cost Function is outside the defined range for some elements in your network, the unit cost used will be the cost of the minimum or maximum value of the attribute you defined in the table.**

### General Section

This section contains general information identifying the Unit Cost Function, as follows:

- **Label** - Unique name that identifies your Unit Cost Function.
- **Element Type** - Displays the type of element to which the function applies, which is always pressure pipe in WaterCAD, but could also be gravity pipe, junction, inlet, manhole, or junction chamber in SewerCAD or StormCAD.
- **Attribute Label** - Element attribute that controls the unit cost, such as pipe diameter. This attribute is selected when you add a new function in the **Unit Cost Function Manager**.

### Attribute Value Range Section

This section displays the minimum and maximum values for the attribute that controls the unit cost in your current network. Click the **Initialize Range** button to have these values calculated.

### Unit Cost Data Table

This allows you to define the Unit Cost Function in a tabular format, preferably defining the costs associated with the entire range of values present in your network. To display the current range of values in your model, initialize the **Attribute Value Range** section by clicking the **Initialize Range** button.

## 10.3.3 Formula Unit Cost Function

The data defining formula-based Unit Cost Functions is grouped as follows:

- **General** - General information identifying the Unit Cost Function.
- **Valid Cost Data Range** - The range for which the function is valid for the attribute used to define the Unit Cost Function.

- **Coefficients** - Coefficients defining the formula relating the unit cost to the attribute value.

In order to help you enter and visualize the function, use one of the following buttons at the bottom of the dialog:

- **Plot** - A graph of the Unit Cost Function.
- **Initialize Range** - The minimum and maximum values of the attribute used to define the Unit Cost Function based on all the elements in your project.

**If the function is invalid for any interval within the Valid Cost Data Range, it is set to 0.0 in that interval. Click the Plot button to see if there are any problems with the function.**

**If the attribute you have selected to define the Unit Cost Function is outside the Valid Cost Data Range for any element in the network, the formula will still be applied to calculate that element unit cost. However, an error message for that element will be reported when computing the cost for the system.**

#### General Section

This section contains general information identifying the Unit Cost Function, as follows:

- **Label** - Unique name that identifies your Unit Cost Function.
- **Element Type** - Displays the type of element to which the function applies, which is always pressure pipe in WaterCAD, but could also be gravity pipe, junction, inlet, manhole, or junction chamber in SewerCAD or StormCAD.
- **Attribute Label** - Element attribute that controls the unit cost, such as pipe diameter. This attribute is selected when you add a new function in the **Unit Cost Function Manager**.
- **Local Unit** - Unit of the attribute that controls the unit cost. This unit is used for defining the formula coefficients.

#### Valid Cost Data Range Section

This section specifies the range of values for which the function is valid for the attribute used to define the Unit Cost Function. Clicking the **Initialize Range** button accesses the two values based on the range of values present in your current network.

#### Coefficients Section

In this section you can enter the coefficients defining the Unit Cost Function. The x-parameter, which represents the value of the attribute on which the Unit Cost Function is based, is expressed by the unit specified in the **Local Unit** field on this tab.

### 10.3.4 Unit Cost Function Notes

In this section you can enter optional notes related to the Unit Cost Function.

## 10.4 Cost Alternatives Manager

The **Capital Cost Alternatives Manager** allows you to edit, create, and manage your cost alternatives. It also gives you more advanced capabilities, such as merging alternatives and creating child alternatives.

On the right side of the dialog are a number of buttons that provide functions for managing the alternatives. These buttons are identical to the buttons found in the **Alternatives Manager**. See the **Alternatives Manager** topic in the Scenarios and Alternatives chapter for a description of the function of these buttons.

To access the **Capital Cost Alternatives Manager,** select **Analysis / Alternatives** from the pull-down menu and select the **Cost** tab, or click the **Cost Alternatives** button in the **Cost Manager**.

# 10.5 Cost Reports

In addition to the standard reporting capabilities, the cost analysis feature provides a number of specialized reports for presenting results. These reports include:

- **Element Detailed Cost Report** - Presents a detailed view of all the cost information entered for a single element.
- **Project Detailed Cost Report** - Provides a detailed view of calculated cost data for every element included in the cost analysis.
- **Project Element Summary Cost Report** - Returns a summary of the costs for every element included in the cost report.
- **Project Summary Cost Report** - Provides an overview of all the costs in the system.
- **Pipe Costs Report** - Provides an overview of the costs associated with the pipes in the project, grouping them by material and section size.
- **Cost Warnings Report** - Provides a list of warnings for a particular Cost Scenario.

Each of these tabular reports can be sent directly to the printer, or copied and pasted into your favorite spreadsheet program for further refinement.

## 10.5.1 Element Detailed Cost Report

This tabular report contains a detailed view of all the cost information entered for a single element. It includes an itemized list of all the construction and non-construction costs for an element, as well as the subtotals and total cost of the element. This report is only available for elements that have been selected for inclusion in the cost calculation.

See the **Tabular Report Window** for information about exporting and printing the data in this report.

To access the field Element Detailed Cost Report**:**

*Stand-Alone:* *Double-click the element for which you wish to see the report, or right-click the element and select* **Edit** *from the drop-down menu. In the dialog that appears, click the* **Report** *button and select* **Cost Report** *from the menu.*

*AutoCAD R14:* *Pick the* **Select** *tool and click the element you wish to edit. If the* **Right-Click Context Menu** *option is enabled, you can also right-click the element and select* **Edit** *from the drop-down menu. In the dialog that appears, click the* **Report** *button and select* **Cost Report** *from the menu.*

*AutoCAD 2000:* *Pick the* **Select** *tool and click the element you wish to edit, or select the element and choose* **Edit** *from the drop-down menu. In the dialog that appears, click the* **Report** *button and select* **Cost Report** *from the menu.*

*AutoCAD 2000i:* *Double-click the element for which you wish to see the report, or right-click the element and select* **Edit** *from the drop-down menu. In the dialog that appears, click the* **Report** *button and select* **Cost Report** *from the menu.*

### 10.5.2 Project Detailed Cost Report

This tabular report contains a detailed view of all the cost information entered for every element included in the cost analysis. It includes an itemized list of all the construction and non-construction costs for each element, as well as the cost adjustments made to the total cost of the project. This report is only available after the costs have been computed for the scenario.

See the **Tabular Report Window** for information about exporting and printing the data in this report.

### 10.5.3 Project Element Summary Cost Report

This tabular report provides a summary view of all the cost information entered for each element selected for inclusion in the cost analysis. It contains an overview of the costs assigned to each element, and an itemized list of the cost adjustments. This report is only available after the costs have been computed for the scenario.

See the **Tabular Report Window** for information about exporting and printing the data in this report.

### 10.5.4 Project Summary Cost Report

This tabular report provides a summary view of all the cost information entered for the elements selected for inclusion in the cost analysis. This report contains an overview of the costs assigned to each element type and an itemized list of the cost adjustments. This report is only available after the costs have been computed for the scenario.

See the **Tabular Report Window** for information about exporting and printing the data in this report.

### 10.5.5 Pipe Costs Report

This printed report provides a summary of the cost of all the pipes included in the cost analysis. The pipes are broken down by material and section size. The total length of pipe for each size and material are reported along with the total cost associated with that group of pipes.

### 10.5.6 Cost Warnings Report

This report provides a list of all the cost warnings for the selected scenario. You will receive warnings when you have assigned a Unit Cost Function to a particular element but the attribute value for that element lies outside of the valid range of data you set for the Unit Cost Function. You need to check these elements and make sure that the cost data supplied to these elements is applicable.

To open the **Cost Warnings Report**, open the **Cost Manager** by selecting **Analysis / Compute Costs** from the pull-down menu, or click the  button on the **Analysis** toolbar. In the dialog that appears, click the **Cost Reports** button and select **Warnings** from the drop-down menu.

Notes

# Chapter 11

# Presenting Your Results

## 11.1 Overview

This chapter covers the various methods that are provided for viewing, annotating, graphing, and reporting your data and results. It also presents the tools available for generating profiles, and color coding elements based on any assigned or calculated attribute.

## 11.2 Element Annotation

Element annotations allow you to display detailed information such as pipe lengths or node ground elevations, as well as calculated values such as velocity, in your drawing. You can add one or more annotations for any type of element in the system. Annotations update automatically. For example, annotations will display newly calculated values and will be refreshed as you change scenarios.

**The annotations and their format are defined by using the Annotation Wizard. In Stand-Alone mode, the annotation format can also be easily modified in the Attribute Annotation dialog, which opens when you double-click the handle of the annotation text in the drawing, or if you right click on the annotation and select Edit <Attribute> from the pop-up menu.**

**Pipe annotations can be aligned with the pipes or displayed horizontally, depending on the Pipe Text alignment setting specified in the Drawing Options dialog.**

You can flip the text from one side of the pipe to the other (reading in the opposite direction) to maintain readability when the pipe direction on a plot is nearly vertical. By default, the text flips direction when the pipe direction is 1.5 degrees measured counter-clockwise from the vertical. You can modify this value by inserting a **TextFlipAngle** variable in the Haestad.ini file, located in the Haestad directory. The angle is measured in degrees, counter-clockwise from the vertical.

For instance if you want the text to flip when the pipe direction is vertical, you should add the following line to the Haestad.ini file:

TextFlipAngle=0.0

Reasonable values fall in the range from 15.0 degrees to -15.0 degrees.

The TextFlipAngle is only applicable to annotations on the plan view.

### 11.2.1 Attribute Annotation Dialog

To access the **Attribute Annotation** dialog right click on the annotation and select **<Attribute Name> Annotation** from the pull-down menu. Alternatively, in the main view in Stand-Alone mode, double-click the handle of the annotation text to display the corresponding **Attribute Annotation** dialog. Here you can easily modify the format of that attribute annotation without going through the **Annotation Wizard** again. The replaceable parameters "%v" and "%u" represent the attribute's value and unit respectively.

### 11.2.2 Annotation Properties

You can access the annotation properties for **Profile Annotations** by right clicking on an annotation and selecting **Annotation Properties** from the pop-up menu. The **Annotation Properties** dialog allows you to adjust the orientation of the text as it appears on the drawing screen and the profile window.

You can edit the following fields within the dialog:

- **Orientation** - Set the orientation of the text in relation to the horizontal axis. You can select from **Horizontal, Vertical,** and **Other** orientations. **Other** is any orientation with a rotation angle not 0 or 90 degrees.
- **Rotation** - Specify the angle in degrees of the text with the horizontal axis.
- **Justification** - Select how the text is justified from either: **Left**, **Right**, or **Centered**.
- **Leader** - Check this box to create a line to be drawn from the text annotation to the item to which it refers.
- **Show Leader Arrow** - Check this box to create an arrow on the leader line from the text annotation.

Instead of utilizing the above options you can also choose to **Align Text with Pipes** for the link annotations.

> **Annotation Properties can be setup within a Profile Template to eliminate the need of reestablishing the same properties every time a new profile is created.**
>
> **These properties are not available for Plan Annotations.**

### 11.2.3 The Annotation Wizard

You can use the **Annotation Wizard** to add annotations to the drawing, as well as to remove or modify existing annotations in the drawing. You can annotate all elements or any subset of elements.

The wizard is divided into three steps:

- **Select Elements** - Select the types of elements to annotate.
- **Specify Annotations** - Specify the set of elements to annotate, the attributes you would like to annotate, and the format of your notations.
- **Summary** - Summary of the selected annotation settings.

To access the **Annotation Wizard**, click the **Annotation Tool** on the toolbar, or select **Tools / Element Annotation** from the pull-down menu.

#### Annotation Wizard - Select Elements

This step allows you to specify the types of elements you wish to annotate by checking the appropriate boxes. You may annotate more than one type of element at a time by checking all the desired element types. If you have already annotated your drawing, you can remove annotations for a particular type of element by unchecking the corresponding box.

> **If you decide to turn off the annotations for a particular element type, your annotation settings will be retained, allowing you to easily toggle annotation back on.**

### Annotation Wizard - Specify Annotation

The next step(s) allows you to specify the subset of elements and the attributes you wish to annotate for each element type. For each element type, you will be presented with a table where you can specify the attributes you wish to annotate, and the mask for each attribute.

- **Specify the set of elements…** - Choose **All Elements** from the choice list for annotation to applied to all elements in the network, or choose a selection set. Click the **ellipsis (…)** button to access the **Selection Set Manager** to edit or add selection sets.
- **Attributes** - Select from a list of all available attributes for the current element type including calculated values. Click this field, and choose the attributes you wish to annotate by selecting from the list that appears. Clicking the sideways triangle button will open the categorized Quick Attribute Selector.
- **Mask** - Customize the way the annotation is displayed. The replaceable parameters "%v" and "%u" represent the attribute's value and unit respectively. By default, the mask is set up as follows: <attribute name>: %v %u.
- **Initial Placement** (Profile Annotation Only) **-** Specify how the annotation will be offset relative to the label of the element it is referring to. If the **<default>** is selected then the annotation will be placed under the label. To specify a custom offset select **Offset…** from the pull-down menu and click the **ellipses (…)** button to bring up the **Offset** dialog. In this dialog you can specify the x and y offsets. Click **OK** when completed.

**When annotating, for example, pipe diameters, the default mask is "Diameter: %v %u". The default annotation for a 150 millimeter pipe would be "Diameter: 150 mm". By changing the mask to "%v %u", the resulting annotation would be "150 mm".**

### Annotation Wizard - Summary

The last step of annotating your drawing is reviewing the choices you have made. If you would like to make changes at this time, simply click the **Back** button to return to previous windows in the wizard. When you are satisfied, click the **Finished** button to apply the annotations to the drawing.

**You can turn annotation visibility on or off by editing the Drawing Options. Your annotation settings will be retained.**

**If the Drawing Options are set so that element annotations will not be displayed, clicking the Finished button in the Annotation Wizard will automatically turn annotations on.**

**In Stand-Alone mode, you can double-click an annotation element in the drawing to edit the associated mask.**

**The Annotation Text Height can be adjusted from the Drawing tab of the Options dialog, accessed by selecting Tools / Options from the pull-down menus.**

## 11.3 Color Coding

Color Coding allows you to assign colors to elements in the drawing based on a variety of input and output attributes. For any attribute, you can supply a color scheme or have the application generate one for you. For example, you can supply a color scheme to display all pipes sizes between 2" and 8" in green, those between 10" and 24" in blue, and those between 27" and 48" in red.

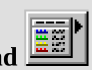
To access Color Coding, select **Tools / Color Coding...** from the pull-down menus, click the **Color Coding** button  on the main toolbar, or double-click a color coding legend figure in the drawing.

### 11.3.1 Color Coding Dialog

At the top of the **Color Coding** dialog are two tabs, **Link** and **Node**. You can set up color coding for both links and nodes, or just one of the two. The following fields are available:

- **Attribute** - Select the attribute by which you would like to color code, or select **<None>** to turn color coding off. By clicking the sideways triangle button you can access the categorized Quick Attribute Selector.
- **Selection Set** - Choose **All Elements** from the pull-down list to apply color coding to be applied to all elements in the network, or choose a selection set to apply color coding to a subset. Click the **ellipsis (…)** button to access the **Selection Set Manager** to edit or add selection sets.
- **Calculate Range** - Automatically determine the minimum and maximum for the specified attribute and selection set.
- **Minimum / Maximum** - Displays the calculated minimum and maximum values for the specified attribute in the selection set.
- **Initialize** - Automatically calculate a default color coding range for the specified attribute, based on the values in your project.

Use the **Initialize** and/or the **Insert** buttons to define your color coding map. Then click **OK** to apply the specified colors to the appropriate elements.

**Color coding legends can be added to any location in the drawing by clicking the Legend**  **button on the Tool Palette.**

**Color coding will automatically update as input or results change. For example, after performing a calculation, colors will update to reflect the newly calculated values.**

**If the results for the selected attribute are not available, or if all values for that attribute are the same, automatic range initialization will not be performed. You can enter your own custom range in this case.**

**A schematic can have any number of color assignments.**

**The Quick View window can be used to display a summary of the active link or node color coding parameters.**

## 11.4 Reporting

### 11.4.1 Predefined Reports

This application provides several predefined reports that can be used in your projects. This feature makes report generation a simple point-and-click exercise. Simply select the elements for which you want a report and send them to your printer.

The following types of Predefined Reports are available:

- Element Details Reports
- Element Results Reports
- Tabular Reports
- Scenario Summary Reports
- Project Inventory Reports
- Plan View Reports

**Detailed reports can be copied to the Windows clipboard in RTF format for use in your favorite word processing program. Refer to the Print Preview window for more information.**

To access the Predefined Reports, select **Report** from the pull-down menus and select the report of your choice.

### 11.4.2 Element Details Report

The **Detailed Reports** dialog allows you to print detailed reports for all elements or any subset of elements in the system.

In Stand-Alone mode, from the **Detailed Reports** dialog, select multiple elements to be printed by holding down the **Shift** key or the **Control** key while clicking with the mouse. Holding down the **Shift** key will provide group selection behavior. Alternately, use the **Select** button to open the **Selection Set** dialog. This provides more powerful selection functions. When you are satisfied, click the **Print** button to output the selected reports.

**You can graphically select elements that you would like to print before opening the Detailed Reports dialog. This is done by holding down the Shift key and selecting elements, or by dragging a window around the area of interest. The selected elements will be highlighted in the list of elements to print when you open the dialog.**

**You can print a detailed report for a single element without using the Detailed Reports dialog. Simply open the element editor for the desired element and click the Report button.**

In AutoCAD mode, to activate the **Detailed Reports** dialog, select **Report / Element Details** from the pull-down menus. The cursor will change to a pick box, signaling you to choose the elements for which you would like to view reports. Select elements as you normally would in AutoCAD. Press the **Enter** key, and the dialog will appear. While all of the elements in the project are listed, the ones you have selected are highlighted. You can use the **Select** button to further edit this list. Click the **Print** button to output the selected reports when you are satisfied.

To access the **Detailed Reports** dialog, select **Report / Element Details** from the pull-down menus.

### 11.4.3 Element Results Report

The **Element Results** dialog allows you to print or preview a single report containing the results for any number of elements in the system.

From the **Element Results** dialog, you can select elements to be printed by holding down the **Shift** key or the **Control** key when clicking with the mouse. Holding down the **Shift** key will provide group selection behavior. Alternately, use the **Select** button to open the **Selection Set** dialog. This provides more powerful selection functions. When you are satisfied, click the **Preview** button to view the selected reports, or click the **Print** button to print the selected reports.

**You can graphically select elements that you would like to print before opening the Element Results dialog. This is done by holding down the Shift key and selecting multiple elements, or by dragging a window around the area of interest. The selected elements will be highlighted in the list of elements to print when you open the dialog.**

**When working with large systems, the preview option can require a great deal of system resources. You can reduce resource requirements by selecting a small subset of elements with which to work. The print option has lower system resource requirements than the preview option.**

In AutoCAD mode, to activate the **Element Results** dialog, select **Report / Element Results** from the pull-down menus. The cursor will change to a pick box, signaling you to choose the elements for which you would like to view reports. Select elements as you normally would select them in AutoCAD. Press the **Enter** key, and the dialog will appear. While all of the elements in the project are listed, the ones you have selected are highlighted. You can use the **Select** button to further edit this list. Click the **Print** button to output the selected reports when you are satisfied.

To access the **Element Results** dialog, select **Report / Element Results** from the pull-down menus.

### 11.4.4 Tabular Reports

Using the powerful FlexTables feature you can very quickly generate a tabular report containing any attribute and any network element.

**All tabular data in this program can be copied to the Windows Clipboard by right-clicking the desired table and selecting Copy from the pop-up menu. You can then paste this data into your favorite spreadsheet or word processor to generate custom reports and graphs.**

To access the **Table Manager**, click the **Tabular Reports** button on the main toolbar or choose **Report / Tables** from the pull-down menus.

### 11.4.5 Scenario Summary Report

The **Scenario Summary** provides a detailed report of the active scenario, including alternatives, and a brief summary of the calculation options.

To access the **Scenario Summary Report**, select **Report / Scenario Summary** from the pull-down menus.

### 11.4.6 Project Inventory Report

The **Project Inventory** report provides a detailed report that includes a summary of the active scenario, a network inventory, and a detailed pipe inventory (grouped by pipe section).

To access the Project Inventory Report, select **Report / Project Inventory** from the pull-down menus.

### 11.4.7 Plan View Report

Generate reports for the plan view of the network, for either the current drawing display (Current View) or the entire drawing extents (Full View).

To generate a preview of the report for the current view or the entire network, choose either **Report / Plan View / Current View** or **Report / Plan View / Full View**, respectively, from the pull-down menus.

### 11.4.8 Calculation / Problem Summary Report

After running hydraulic calculations, the **Results** tab of the **Scenario Editor** is displayed. This tab contains a summary of the calculation results. To view any problems or warnings encountered during the simulation click the **Element Messages** button.

> **Holding the mouse cursor over a message in the Element Calculation Message Browser report will open up a pop-up message box that displays further information on the message.**

The report consists of a series of folders that represent different stages of the calculation process. Double click a folder or click the **+** sign to view information related to the folder's caption. The color of the folder will indicate if any problems occurred during that portion of the analysis.

The colors indicate the following:

- **Green light** - Calculations were run successfully, without any warning or error messages being generated.
- **Yellow light** - Calculations were run successfully, without error messages being generated. However, there are one or more warning messages. Warnings are displayed in the results summary in this tab.
- **Red light** - Calculations were not run successfully and error messages were generated, as shown in the results summary of this tab.

This report can be previewed before being printed or copied to the clipboard by clicking the **Printer** button on the **Results** tab. It can also be exported to a text file by clicking the **Save** button on the **Results** tab. Only the exposed text will be exported, copied or printed.

The **Results** tab is accessed from the **Scenario Manager**, reporting the results corresponding to the highlighted scenario in the scenario tree view. It can also be accessed for the currently active scenario by clicking the **GO** button, which opens the **Scenario Editor**.

## 11.5 Graphing

### 11.5.1 Graph Setup

The **Graph Setup** dialog allows you to view and compare data for time-based attributes for multiple elements over multiple scenarios within the hydraulic network. It is accessed either when you select **Element Graph / <Element Type>** or **Hydrograph** from the **Report** pull-down menu.

The **Graph Setup** dialog is divided into three or two tabs depending on whether you selected **Hydrograph Report**.

- **Graph Setup -** From this tab you can select the dependent variable to be graphed against time. This tab is not available when graphing hydrographs, as flow is automatically the dependent variable.
- **Elements** - From this tab you can select which elements to include in the graph. There are three ways to select these elements:
  - You can check each of the elements you wish to include in the graph in the **Available Elements** list.
  - You can click the **Select** button and choose the elements by using the **Selection Set** dialog.

- You can graphically select elements that you would like to print before opening the **Graph Setup** dialog. In Stand-Alone mode, holding down the **Shift** key allows you select multiple elements. You can also select elements by dragging a window around the area of interest. The selected elements will be highlighted in the list of elements for graphing when you open the dialog, when you open the **Graph Setup** dialog through the **Reports** menu.

- **Scenarios** - From this tab you can select which scenarios to use when graphs are generated from a list of scenarios with calculated EPS results.

### 11.5.2 Graph Window

The **Graph** window is divided into two tabs: **Graph** and **Data**. The **Graph** tab displays a plot of the selected dependent variable vs. time. The **Data** tab will display the data under the **Graph** tab in a tabular format.

The following functions will either be formed on the plot or the tabular data depending on which tab is selected.

- **Copy** - Copies the graph / data onto the Windows Clipboard for use in other applications.
- **Print** - Outputs the contents of the **Graph/Data** tab to the printer.
- **Options / Graph Options** - Allows you to customize the plot by changing the graph's axes, fonts, titles, etc. This is only available when the **Graph** tab is selected.
- **Options / Graph Setup** - Allows you to rebuild the graph with different data and parameters.
- **Close** - Close the **Graph** window.
- **Help** - Provides access to help for the **Graph** window.

The graph window is accessible whenever you plot time-based data through the **Graph Setup** dialog.

### 11.5.3 Plot Window

The **Plot** window provides the following functionality:

- **Copy** - Copies the plot onto the Windows Clipboard for use in other applications.
- **Print** - Outputs the contents of the **Plot** window to the printer.
- **Options / Graph Options** - Allows you to customize the plot by changing the graph's axes, fonts, titles, etc.
- **Close** - Close the **Plot** window.
- **Help** - Provides access to help for the **Plot** window.

The plot window is accessed whenever you generate non-time based plots such as cost function curves or pump curves, or whenever you plot input data such as loading patterns.

### 11.5.4 Graph Options

These features allow you to customize the way a graph or pie chart looks. The dialog is divided into several tabs:

*Titles*

- **Titles** - There are three sets of titles for a graph: Graph title, X-Axis title and Y-Axis title. Each title set contains two levels: title and subtitle. A pie chart simply has a title and a subtitle.

- **Title Font** - This feature allows you to select and change the text font type for specific items on the graph or pie chart. Use the selection list to choose the item for which to change the font, then click the **ellipsis (…)** button to select the desired font type from the list of available fonts currently installed on your PC.

### *Axis (for graphs only)*

- **Automatic Scaling** - By default, the program uses the Automatic Scaling options for setting the X and Y-axis minimum, maximum, and increment values. To customize an axis, turn the check mark off and enter the desired values for the minimum, maximum, and increment. If desired, you can customize a single axis while leaving the other in the Automatic Scaling mode.
- **Log Scale** - Place a check mark in this box to use a log scale for this axis. You can use a log scale for one or both axes.

### *Grid (for graphs only)*

- **X-Axis** - Place a check mark in this box to view grid lines corresponding to the X-Axis labels.
- **Y-Axis** - Place a check mark in this box to view grid lines corresponding to the Y-Axis labels.
- **Line Color** - Use this selection list to define the color to use for both axes grid lines.
- **Line Style** - Use this selection list to define the line type (solid, dashed, etc) to use for both axes grid lines.
- **Fill Color** - Use this selection list to define the color to use for background fill within the plotting boundaries of the graph.
- **Save as Default** - Place a check mark in this box to save the current grid settings as the default for subsequent graphs.

> **You can specify to use grid lines for one or both axes.**

### *Display (for pie charts only)*

- **Data Labels** - Allows you to annotate pie charts with percentages, labels, or both.
- **Percentages** - Indicates how many decimals are to be displayed for the percentage figures.
- **Legend Location** - Allows you to place the legend (if any) on the left, right, top, or bottom of the pie chart.
- **Chart View** - Allows you to generate a 3D-view pie chart.

### *Legend*

- **Show Legend** - A check mark designates that the legend will be included on the graph or pie chart. Turn the check mark off if you do not wish to show the legend.
- **Series** - Each series represents a different curve on the graph or a slice on the pie chart. If the graph contains only one curve, or the pie chart contains only one slice, then it is designated as Series 1. Scroll through the list and select the desired curve or slice (series number). Then, use one of the options below to customize it:
  - **Label** - Name for the selected curve (series).
  - **Line Color** - Color for the selected curve (series).
  - **Line Style (for graphs only)** - Style for the selected curve (series).
  - **Line Width (for graphs only)** - Width for the selected curve (series).

  – **Symbol (for graphs only)** - Data point symbol to use for the selected curve (series).

- **Save as Default** - Place a check mark in this box to save the current legend settings as the default for subsequent graphs.

Access the **Graph Options** by clicking the **Options** button at the top of a **Plot** or a **Graph** Window.

## 11.6 Pie Charts

Pie charts can be generated for the following, at any node:

- **Base Load** - Shows the components of the sanitary loads entered at a node. This type of pie chart is available for manholes, junction chambers, pressure junctions, and wet wells.
- **Sanitary (Dry Weather) Flow** - Shows the components of the system sanitary flow at a node.
- **System Flow Summary** - Shows the components of the total flow at any gravity node.
- **Total Cost** - Shows the components of the total costs for a given scenario.

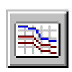
You can create a **Sanitary Flow** or **System Flow Summary** pie chart after calculations have been run. Open a node's **Element Editor**, click the **Report** button, point to the **Pie Chart** option, and then select the desired option.

The **Base Load** pie chart is obtained by opening a node's **Element Editor**, selecting the **Loading** tab, and clicking the **Pie Chart** button at the right.

You can create a **Total Cost** pie chart by opening the **Cost Manager**, right-clicking the desired scenario, and selecting the **Graph** option.

## 11.7 Profile

A profile is a graphical cross-section of a portion of your sewer system. The **Profile** view shows information such as ground elevation, pipe inverts, crowns, sumps, and hydraulic grade.

To access Profiling, click the **Profile** button  on the main toolbar, or choose **Tools / Profiling**… from the pull-down menus, or right-click the element from which you would like to begin profiling.

### 11.7.1 Profile Manager

Within the **Profiles Manager**, profile views can be created and accessed. The previously created profile views will appear in the main pane of the dialog. To view a profile in the list, select it and click the **Open** button.

By clicking the **Profile Management** button the following functions can be performed:

- **Add** - Creates a new profile view.
- **Edit** - Select elements to be profiled and edit the **Profile Options** for the highlighted profile.
- **Rename** - Renames the selected profile.
- **Delete** - Deletes the selected profile.
- **Notes** - View and edit notes associated with the selected profile.
- **Apply Template** - Update a profile based on a new (or revised) template.

Click the **Templates** button to open up the **Profile Templates** dialog.

## 11.7.2 Profile Templates

### Profile Templates Dialog

The **Profile Templates** dialog allows you to create and modify pre-defined templates that can be applied to new profiles in the first step of the **Profile Wizard**. The templates will then establish the annotations and text properties for the new profile, instead of having to reestablish all the information every time you create a new profile.

The following buttons are displayed on the dialog:

- **Add** - Create a new template. When this button is clicked you will be prompted to name the template. Click **OK** to continue to the main **Template** dialog.
- **Edit** - Modify the existing template selected from the list.
- **Duplicate** - Create a copy of the selected template. When this button is clicked you will be prompted to name the duplicate.
- **Delete** - Remove the selected template from the list.
- **Rename** - Change the name of the selected template. When this button is clicked you will be prompted for the new name.
- **File** - Import and Export templates for use by other users.

To access the **Profile Templates** dialog click the **Templates** button on the **Profiles** dialog.

### Templates

A template is a predefined set of text styles and annotations that can be applied to a new profile upon its creation. By setting up a single template, you can eliminate having to re-setup the generic portions of the profile every time a new one is created.

The **Template** dialog is divided into five tabs:

- **Layout Options** - Edit the configuration of the profile text.
- **Drawing** - Edit the drawing scale and the axes labeling
- **Annotation** - View the existing annotation and modify it.
- **Labels** - Edit the profile labels.
- **Layers** - Set up the layer configurations for the template.

### Template Layout Options Tab

Under the **Layout Options** tab you set up the configuration for the pipe and node text for the profile template. The following options are identical for both node and pipe elements:

- **Orientation** - Set the orientation of the text in relation to the horizontal axis. You can select from **Horizontal, Vertical,** and **Other** orientations. **Other** is any orientation with a rotation angle not 0 or 90 degrees.
- **Rotation** - Specify the angle in degrees of the text with respect to the horizontal axis.
- **Justification** - Select how the text is justified from either: **Left**, **Right**, or **Centered**.
- **Leader** - Check this box to create a line from the text annotation to the item to which it refers.
- **Show Leader Arrow** - Check this box to create an arrow that will appear on the leader line from the text annotation.

Instead of utilizing the above options you can also choose to **Align Text with Pipes** for the pipe annotations.

> **These same options are available for individual annotations on the Profile Window by right clicking on one of the annotations and selecting Annotation Properties from the pop-up menu.**

### Template Drawing Tab

Under the **Drawing** tab you can set various options associated with the drawing scale and axes labeling to be specified in the template. The **Drawing** tab of the **Template** dialog has the following sections:

- **Drawing Scale** - In this section you can choose to set the scale based on the project scale by checking the **Match Project** check box. If the box is unchecked then you can specify your own horizontal and vertical scales as well as the default text height multiplier.
- **Axis** - In this section you can specify the horizontal and vertical increment for the axis labeling, as well as the position of the elevations on the Y-axis. For example, if you choose **Left** from the **Axis Labeling** drop-down menu then labels will appear on the left side of the profile. You can choose for the labels to also appear on the right side of the profile or on both sides of the profile.

### Template Annotation Tab

Under the **Annotation** tab you can view the annotations selected for the current template, as well as each annotation's initial placement.

To open the **Annotation Wizard** to edit the template annotations click the **Edit** button.

### Template Labels Tab

Under the **Labels** tab you can specify various labels associated with the profile, as well as an associated height multiplier.

In the label fields you can put one of several dynamic parameters that will automatically update as the model updates. The following table shows the list of available parameters and an example of their use.

| Dynamic Parameter | Function | Label Input Example | Label Output Example |
|---|---|---|---|
| %s | Displays the name of the current scenario in the label. | Scenario: %s | Scenario: Base |
| %p | Displays the profile name. | Profile: %p | Profile: Trunk Line |
| %t | Displays the current time step for the scenario. | Scenario: %s (%t) | Scenario: Base (0.00 hr) |
| %u | Displays the units of the appropriate attribute. | Station (%u) | Station (ft) |

### Template Layers Tab

From the **Layers** tab you can set various properties for the different layers associated with a hydraulic profile. You can click whether or not the layer is visible. You can also set the color for the layer. If you click the **ellipses (…)** button from inside one of the **Color** fields you can select or create a customized color.

The properties established for the layers will remain if the profile is exported to a DXF or directly to AutoCAD.

### Sharing Templates Between Projects

All template information created at a computer is stored in the file called ProfileTemplates.LTM, located in the product directory. So if you create several templates types that you wish to share between different users, simply transfer the file to the other machines with the product installed, and copy over the existing profile templates file.

Alternatively, you can export individual profile templates for import by other users by clicking the **File** button on the **Template Manager** dialog and selecting **Export Template** from the pop-up menu. This creates an .LTF file with a saved copy of the selected template.

To import the template, select **Import Template** after clicking the **File** button in the **Template Manager**. This will copy the template data to the ProfileTemplates.LTM file on that machine for reuse.

## 11.7.3 Profile Wizard

The **Profile Wizard** is provided to guide you through the profile generation process. You will be asked to specify a profile start and stop node, as well as the horizontal and vertical axis scaling parameters.

### *Profile Wizard - Profile Elements*

The first step of the **Profile Wizard** allows you to specify the elements included in the hydraulic profile. The specified elements will appear in the list pane in the dialog. You can modify the list by clicking the following buttons:

- **Select From Drawing** - Clicking this button will open the plan view of the hydraulic network. Simply click the elements you wish to include in the profile. The profile must consist of a single path that can go from upstream to downstream or downstream to upstream or in both directions. The elements can include both gravity and pressure elements. Right click and select done when finished. In AutoCAD mode simply click the drawing pane to return to the **Profile Wizard**.
- **Remove All** - This will delete all the elements from the profile list.
- **Remove All Previous -** This will delete all elements above the selected element in the list.
- **Remove All Following -** This will delete all the elements below the selected element in the list.

In the first step of the **Profile Wizard**, you can also establish user-defined stationing for the node elements in the profile, and weather the calculated **Stationing Order** is increasing or decreasing.

To establish a user-defined station for a node in the list, click the **User Defined Station** checkbox in the row occupied by the node. All stationing upstream of that node will be recalculated based on the entered station if they are not specified as user defined.

### *Profile Wizard - Templates and Notes*

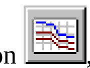
The second step of the **Profile Wizard** allows you to specify the template to be applied to the new profile. Select it from the **Template** pull-down menu. Click the **ellipses (…)** button next to the **Template** pull-down menu to open up the **Profile Templates** dialog. You can also enter any notes about the profile in the large text box below the notes icon.

**Templates are only used initially when defining the profile. Once the profile is created all connections to the selected template are severed. Therefore if you change a characteristic of the template the change will not be reflected in the profile. This prevents undo modification.**

**You can reapply a template through the Profiles dialog. Click the Profile Management button and select Apply Template.**

#### *Profile Wizard - Options*

The third step of the **Profile Wizard** allows you to customize the horizontal and vertical axis scaling parameters. You can choose either automatic or user-defined axis scaling. Use automatic scaling to accept the default scale for the profile axis. Choose user-defined scaling to specify the axis minimum, and maximum values. The increment value is accessible regardless of which option is selected. You can also set the direction of the profile either from left to right or from right to left.

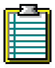
The **Profile Wizard** can be accessed by clicking the **Profile** button , and then clicking the **Profile Management** button and selecting add. You can also right-click the node from which you would like to begin profiling, and select **Profile** from the pop-up menu. In AutoCAD mode, this requires you to have the **Right-Click Context Menu** toggled **On** in the **Global Options** dialog, which is accessed from **Options / Global Options**.

### 11.7.4 Profile Window

The profile that you specified will be displayed in the **Profile Plot** window. The following functions are available in the main toolbar.

- **File / Export to Drawing** - (AutoCAD mode only) Export the file to your AutoCAD drawing.
- **File / Export to DXF** - Export the profile in a .DXF format for use in AutoCAD.
- **Undo / Redo** - Click the left facing arrow to undo the previous command. Click the right facing arrow to redo the command.
- **Pan** - Move around in the profile window.
- **Zoom Support** - Zoom in, Zoom Out, Zoom Extents, and Zoom Window.
- **Options -** Opens the menu with the following items
  - **Find Element** - Locate an element in the **Profile Window**.
  - **Profile Options** - Set Axis and Drawing options and various other user-specified parameters.
  - **Annotation Manager** - Set annotations for the elements in the profile.
- **Print Preview** - Preview the **Profile** view.
- **Close** - Exit the **Profile** window. Any edits made in the **Profile** window will be lost when you exit.
- **Help** - Opens the program's Help database.

In addition to the functions on the main toolbar, you can add your own text and graphic annotations to the profile by clicking the representative buttons on the vertical side-toolbar of the **Profile Plot** window.



**You can make adjustments to your profile options after generating the profile by using the Options button at the top of the Profile Window.**

**You can use the mouse to graphically drag and reposition the textual annotations in the Profile window.**

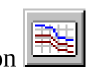
**The Hydraulic and Energy Grade Lines will only be plotted if the calculated results are valid. You may need to Compute your model to make sure the results are valid.**

**Use the scroll bars along the right and bottom of the Profile window to pan left, right, up, and down.**

**Edit the outlet element to change the starting station used in profiling.**

**Use the Scenario control located at the top of the Profile window to see the profiles for different scenarios.**

**Use the VCR Controls to animate the hydraulic grade line for an EPS analysis.**

The **Profile** window is accessed by clicking the P**rofile** button  and opening an existing profile in the **Profiles Manager**. The **Profile Window** will also appear after completing the **Profile Wizard**.

## Profile Options Dialog

The **Profile Options** dialog allows you to customize the horizontal and vertical axis scaling parameters, the line widths, text height, and layer visibility. The dialog is divided into four tabs:

- Elements tab - This tab is only available when you open the profile options by selecting **Profile Management / Edit** from the **Profiles** dialog.
- Axis tab
- Drawing tab
- Layers tab
- Background tab

## Profile Elements Tab

Within the **Elements** tab you can specify the elements to include in the hydraulic profile. The specified elements will appear in the list pane in the dialog. You can modify the list by clicking the following buttons:

- **Select From Drawing** - Clicking this button will open the plan view of the hydraulic network. Simply click the elements you wish to include in the profile. The profile must consist of a single path that can go from upstream to downstream or downstream to upstream or in both directions. The elements can include both gravity and pressure elements. Right click and select done when finished. In AutoCAD mode simply click the drawing pane to return to the **Select Elements** dialog.
- **Remove All** - This will delete all the elements from the profile list.
- **Remove All Previous** - This will delete all elements upstream of the selected node.
- **Remove All Following** - This will delete all the elements downstream of the selected node.

You can also establish user-defined stationing for the node elements in the profile, and weather the calculated **Stationing Order** is increasing or decreasing.

To establish a user-defined station for a node in the list, click the **User Defined Station** checkbox in the row occupied by the node. All stationing upstream of that node will be recalculated based on the entered station if they are not specified as user defined.

To access the **Profile Elements** tab, click the **Profile Management** button in the **Profiles** dialog and click **Edit**. Then click the **Profile Elements** tab.

### Profile Axis Tab

The **Axis** tab allows you to customize the horizontal and vertical axis scaling parameters. You can choose automatic or user-defined axis scaling. Use the automatic scaling to accept the default scale for the profile axis. Choose user-defined scaling to specify the axis minimum, and maximum values by deselecting the **Automatic Scaling** check box. You can also select the position of the labeling on the vertical axis. For example, if you select **Left** from the **Axis Labeling** drop-down menu, the elevation labels on the Y-axis will appear on the left hand side of the profile. You can also choose for the labels to appear on the right side of the profile, as well as on both sides of the profile.

Under the **Axis** tab you can also set the direction of the profile in the **Direction** field

### Profile Drawing Tab

The **Drawing** tab allows you to set line widths for some of the profile figures, toggle layer visibility, and adjust the size of the text in the profile.

To change the line widths for the ground elevation, the node structures, and the hydraulic grade and energy grade lines in the profile, simply enter the desired size in their respective fields.

The **Text Height Annotation Multiplier** allows you to adjust the size of the text in the profile.

You can also adjust the line widths for various graphic attributes presented on the profile such as:

Ground Elevation

Structures

Hydraulic Grade

Energy Grade

### Profile Layers Tab

From the **Profile Layers Tab** you can set various properties for the different layers associated with hydraulic profile. You can click whether or not the layer is visible. You can also set the color for the layer. If you click the **ellipses (…)** button from inside one of the **Color** fields you can select or create a customized color.

The properties established for the layers will remain if the profile is exported to a DXF or directly to AutoCAD.

### Profile Background Tab

A .DXF image can be inserted as a background for the selected profile.

- **DXF Background Filename** - This field enables you to specify a .DXF file to be used as the background for your project. Enter the drive, directory, and file name, or click the **Browse** button to select a file interactively.
- **Show Background** - If the background .DXF file is turned off, it will not be read from a disk or displayed in the drawing pane.

- **DXF Unit** - The .DXF drawing unit conversion is used when importing .DXF background files, and also when exporting a .DXF file from the project. Note that the value in this field governs the import behavior for .DXF files saved in scientific, decimal, or fractional units, but not for .DXF files saved in architectural or engineering units.
- **Insertion Point** - These fields enables you to specify the point (X and Y coordinates) in the profile where the drawing will be inserted.

.DXF file import behavior is governed by specific factors within the .DXF file. If a file does not import as you expect, check the options used to generate it carefully. For example, try importing the .DXF back into the original program or into another program that supports the .DXF format, such as AutoCAD or MicroStation. If the file does not import into other applications, there may be an invalid or missing header, invalid elements, or other errors.

#### Export Profiles

In AutoCAD mode, profiles can be exported to an AutoCAD drawing using the **File** menu on the **Profile Window**. Profiles will be exported to an insertion point below the current drawing extents.

In Stand-Alone mode, profiles can be exported into a .DXF file. After performing a few preliminary steps you can import the file into AutoCAD.

## 11.8 Diversion Network

The **Diversion Network** window provides a graphical view of the diversion links in the system. The flow arrows on the links in the diversion network indicate the directions of diverted flow. The diversion network is displayed with the existing network as the background.

After the model is calculated, diversion links will be annotated with the percent diverted out**.**

### 11.8.1 Diversion Network Window

The diversion network will be displayed in the **Diversion Network** window. The following functions are available in the **Diversion Network** window:

- **File / Export to DXF** - Export the network in a .DXF format for use in AutoCAD.
- **File / Export to Drawing** (AutoCAD mode only) - Export the network into your AutoCAD drawing.
- **Pan** - Moves around the diversion network to change view.
- **Zoom Support** - Zoom in, Zoom Out, Zoom Extents, and Zoom Window.
- **Options** - Toggle background visibility, set background color, and find elements.
- **Print Preview** - Preview the network.
- **Close** - Exit the network window.

**You can specify the diversion target on the Diversion tab of the element's editor.**

**Use the scroll bars along the right and bottom of the Diversion Network window to pan up, down, left, and right.**

**Use the Scenario control located at the top of the Diversion Network window to see the gutter network for different scenarios.**

### 11.8.2 Diversion Network Options

The **Options** button located at the top of the **Diversion Network** window allows you to change the color of the background drawing, toggle the visibility of the background drawing, and find elements.

### 11.8.3 Diversion Network Background Color

This dialog allows you to optionally display your background drawing in a single color. To display the background in a single color, select the desired color from the list. To display the background using the current background drawing colors select **<Auto>**.

## 11.9 Scenario Comparison

The data calculated in different scenarios can be compared through the use of the **Scenario Comparison** window. This allows you to create an annotated drawing to display the differences in the values for any two scenarios.

### 11.9.1 Annotation Comparison Wizard

The **Annotation Comparison Wizard** is used to create a drawing that contains text elements displaying the differences between specific attributes of two scenarios. The **Annotation Comparison Wizard** is identical to the **Annotation Wizard** except it has one additional step. This step involves selecting the two scenarios you wish to compare.

- **Scenario 1** - Choose the baseline scenario.
- **Scenario 2** - Choose the scenario you wish to compare to Scenario 1.

The value in Scenario 1 is subtracted from the value of Scenario 2, and the difference is displayed. Therefore, if any specified attribute's value is greater in Scenario 2 than it is in Scenario 1, the difference is displayed as a positive number. If the value is smaller in Scenario 2 than in Scenario 1, it is displayed as a negative number.

For example, suppose your model contains two scenarios. One is named 2002 Conditions, and the other is named 2010 Conditions. To create a drawing that displays the difference in velocity in a pipe between the 2002 scenario and the 2010 scenario, you would use the **Annotation Comparison Wizard**. You could choose the 2002 scenario as Scenario 1, and the 2010 scenario as Scenario 2. You would then complete the rest of the steps in the wizard. The drawing produced would show positive values where the velocity increased under 2010 conditions and negative values where the velocity decreased under 2010 conditions.

> **For applications that support extended period simulations, you can choose the same scenario for Scenario 1 and Scenario 2 to annotate the differences between two time steps of that scenario.**

To access the **Annotation Comparison Wizard**, open the **Scenario Manager** and click the **Scenario Comparison** button.

### 11.9.2 Scenario Comparison Window

The **Scenario Comparison** window allows you to view, print, export, and modify scenario comparison annotations.

Along the top of the window is a row of buttons that perform the various functions listed below:

- **File / Export To DXF** - Exports the drawing in the standard .DXF file format.

- **File / Export To AutoCAD** (available only in AutoCAD mode) - Export the drawing to the current AutoCAD drawing.
- **Zoom Tools** - Provides standard zoom capabilities for navigating within the drawing.
- **Options / Annotation Manager** - Opens the **Annotation Comparison Wizard** to add, delete, or modify the scenario comparison annotations.
- **Options / Annotation Height Multiplier** - Modifies the text height for the scenario comparison annotations.
- **Options / Find Element** - Allows you to locate an element by its label.
- **Print Preview** - Opens the **Print Preview** window to view how the printed page(s) will look.
- **Close** - Closes the Scenario Comparison window.
- **Help** - Get quick access to this Help topic.

Several user interface elements are available to let you modify the scenarios that are being compared, and to control when the scenario comparison annotations are updated. These interface elements are described in more detail below.

- **Scenario 1** - This row of controls is similar to the **Analysis Toolbar** on the main window. This field allows you to choose, from the list of available scenarios, the one that will be the baseline in the comparison.
- **Scenario 2** - This row of controls is identical to those described above in Scenario 1, but instead of defining the baseline for the comparison, the scenario you pick here will be compared to the baseline.
- **Update** - Click this button to refresh the scenario comparison annotations. This button is used when Auto Update (described below) is off, and you have changed either Scenario 1 or Scenario 2.
- **Auto Update** - A check in this box indicates that **Auto Update** is on, and that the scenario comparison annotations will be refreshed whenever Scenario 1 or Scenario 2 is changed. With Auto Update off, you can select the desired combination of Scenario 1 and Scenario 2, then click the **Update** button. With **Auto Update** on, the annotations will refresh automatically to every scenario or time step change.

The **Scenario Comparison** window is accessed by clicking the **Scenario Comparison** button in the **Scenario Manager**, and then completing the **Annotation Comparison Wizard**.

## 11.10 Graphic Annotation

In Stand-Alone mode, several **Graphic Annotation** tools are provided for enhancing the appearance of your drawing. Graphic annotations can be manipulated like any other element in the **Graphical Editor**. You can add, move, and delete them just as you would with any network elements.

To add graphic annotation to your drawing, use the **Tools\Layout / Graphic Annotation** menu item, or use the tool-palette located along the left side of the main window. The available tools are:

- **Line Tool** - Add polylines or polygons such as drawing roads or catchment outlines.
- **Border Tool** - Add rectangles to your drawing for creating borders such as property lines.
- **Text Tool** - Add text to your drawing for adding explanatory notes, titles, or labels for non-network elements.

**The program will calculate the area of a closed polyline. Right-click the polyline for which you wish to determine the area and select Enclosed Area.**

**To open or close a polyline, right-click the polyline and select Close. A check will appear next to the menu item to indicate that the polyline is closed.**

**To add bends or vertices to a polyline, right-click the polyline at the location you would like to add a bend and select Bend/Add Bend.**

**To remove bends or vertices from a polyline, select the polyline, right-click the bend you would like to remove, and select Bend/Remove Bend.**

### 11.10.1 Legend

Legends are used to display the ranges of the active link and node color coding. The legend tool adds a color coding legend to the drawing. This legend is automatically updated as the color coding is modified.

Editing of the legend figure is not required. In Stand-Alone mode, multiple legends may be placed in the drawing to assist you when printing specified regions within the drawing.

**You can double-click a color coding legend in the drawing to edit the associated color coding parameters.**

## 11.11 Preview Windows

This window provides you with a preview of what will be printed. The window contains the following buttons:

- **Pg Up / Pg Dn** - Navigate between pages of the report.
- **Copy** - Copy the report(s) to the Windows Clipboard.
- **Print** - Output the report to the printer.
- Options
  - ***Print Setup***- Change printer options, such as portrait or landscape page layout.
  - ***Fit to Page*** - The **Fit to Page** check box will not appear if the **Print Preview** window does not contain a drawing, or if the drawing is in schematic mode. When checked, the drawing will be scaled to fit within a single page. When not checked, the drawing will be output using the drawing scale.
- **Close** - Close the **Print Preview** window.
- **Help** - Provides access to help for the **Print Preview** window.

## 11.12 Status Log

Several commands generate a status log showing the results of that command. For instance, a status log is displayed when you calculate a scenario using the **GO** button. The status information is displayed at the top of the dialog. The dialog contains the following buttons:

- **Save** - Export the status log results as an ASCII file.
- **Print or Print Preview** - Print or preview the status log results.
- **Close** - Close the status log dialog after design calculations.
- **Help** - Access context-sensitive online help.

Notes

# Chapter 12

# Engineering Libraries

## 12.1 Engineering Libraries Overview

The Haestad Methods' **Engineering Libraries** and **Library Managers** are powerful and flexible facilities for managing specifications of common materials, objects, or components that are shared across projects. Some examples of objects that are specified through engineering libraries include pipe materials, pipe sections (in StormCAD and SewerCAD), and sanitary loads (in SewerCAD only). You can modify engineering libraries and the objects they contain by using the **Tools / Engineering Libraries** option, or by clicking the **ellipsis (…)** buttons available next to the fields in dialog boxes that make use of library objects.

The data for each engineering library is stored in a tabular ASCII file with the extension .HLB.

**We strongly recommend that you only edit these files using the built-in facilities available by selecting Tools / Engineering Libraries. If absolutely necessary, these library files may be edited or repaired using any ASCII editor.**

The standard set of engineering libraries shipped with your Haestad Methods product reside in the product’s program directory. By default, each project you create will use the objects in these default libraries. In special circumstances, you may wish to create custom libraries to use with one or more projects. You can do this by copying a standard library or creating a new library, and setting the path in the **Engineering Library Manager** to the path for the custom library.

When you change the properties for an object in an engineering library, those changes will affect all projects that use that library object. At the time a project is loaded, all of its engineering library objects are synchronized to the current library. Objects are synchronized based on their label. If the label is the same, then the object’s values will be made the same. If any library referenced in a **Library Manager** path cannot be found at the location specified, then the standard library in the program directory will be used. Once a project is created, it is not necessary to have access to the engineering library in order for that project to be edited or analyzed.

## 12.2 Engineering Library Manager

The **Engineering Library Manager** dialog consists of a table of five columns: **Library**, **Current Path**, **Browse**, **Edit**, and **New**. There is one row for each kind of engineering library used in your project. You cannot create library types different from the set of standard libraries shipped with the product. The columns in the table are as follows:

- **Library -** This column lists the kind of object stored in the referenced library.
- **Current Path -** This column lists the path to the library to be used for objects of a certain kind within the current project. By default, the path will reference the standard library shipped with your Haestad Methods product. To browse for other libraries of the same type that you may have already created, click the **Browse** column.

- **Browse -** Click this column and the button that appears if you wish to search your computer or network and locate other engineering libraries. To reference a library in the path field, the library must already exist. To create it you may copy a standard library using Windows File Manager or Explorer, or click **New** as described below.
- **Edit -** Click this column and the button that appears if you wish to add, delete, or edit the objects within a specific kind of engineering library.
- **New -** Click this column and the button that appears if you wish to create a new library.

**Most users do not need to create custom libraries or edit the library paths. You only need to change path values if you wish to create and use custom libraries.**

The **Engineering Library Manager** can be accessed by selecting **Tools / Engineering Libraries** from the pull-down menus.

## 12.3 Engineering Library Editor

The **Engineering Library** dialogs consist of a table with two columns:

- **Label** - This column contains a textual description of the object. In general, objects are considered to be the same if their labels are the same. For example, when a project is loaded, the engineering library objects are synchronized to the current library based on label.
- **Available in ...** - This column contains a checkbox indicating whether the library object on the given row is enabled for use by this application. If an object is enabled, it will appear in choice lists as a candidate for use in the project. If an object is disabled, it will remain in the library and be editable, but it will not be offered as a candidate for any operations in the program. If a disabled object has already been used in a project, then it will remain in use. Disabling it will not affect the existing project in any way.

The following command buttons appear on the **Engineering Library** dialog:

- **Insert** - Insert a new, unlabeled object into the current library. You must then click the **Edit** button to edit the label and add the appropriate values before the library will be valid. Library objects will be sorted by label in ascending alphabetical order the next time you open the **Engineering Library** dialog.
- **Duplicate** - Create a copy of the currently highlighted object at the bottom of the list.
- **Delete** - Delete the object represented by the highlighted row. Note that this command always deletes objects from the library, but never deletes an object from your current project if it is in use. To change the library object that is currently in use by a project, proceed to the dialog containing the field where the library object is referenced and select a different library object.
- **Edit -** Access the object properties editor.
- **Usage -** Only applies to the material engineering library. Use this button to specify specific uses for the material.

The **Engineering Library Editors** can be accessed by selecting **Tools / Engineering Libraries** from the pull-down menus and clicking the **Edit** column and the button that appears next to the Library you want to edit.

# 12.4 Usage

This dialog only applies to the **Material Library**. Usage is what specifies the type of section or pipe that will be available for each material. Use the following commands to select which sections you would like to be available for each material:

- **[ > ]** Adds the selected item(s) from the **Available Items** list to the **Selected Items** list.
- **[ >> ]** Adds all of the items in the **Available Items** list to the **Selected Items** list.
- **[ < ]** Removes the selected item(s) from the **Selected Items** list.
- **[ << ]** Removes all items from the **Selected Items** list.

# 12.5 Extreme Flow Factor Method Library

Extreme flow factors are generally used for computing peak discharges, and therefore are typically referred to as peaking factors or peaking equations. However, since they can also be used to compute minimum discharges, the term extreme flow factor is more accurate and will be used throughout the program and documentation.

SewerCAD defines tabular and equation extreme flow factor methods in the editable **Engineering Libraries**, thus allowing you to edit predefined methods and insert new ones. The extreme flow factor can be user-defined with either of the following:

- Equation extreme flow factor method
- Table extreme flow factor method

In both cases, the extreme flow factor method can be a function of either of the following:

- Contributing population
- Base Load

> **Discharge based extreme flow methods can be used with any unit dry load. Population based extreme flow methods can be used only with population-based unit sanitary and non-population-based unit sanitary loads that have population equivalents specified.**

## 12.5.1 Extreme Flow Factor Equation Properties

SewerCAD uses a generic exponential equation to define any extreme flow factor method. For population-based extreme flow factor methods, the generic equation is:

$$EFF = c_1 + \frac{c_2 + (m_1 + P)^{e_1}}{c_3 + (m_2 P)^{e_2}}$$

where P is population and c1, c2, c3, m1, m2, e1, and e2, are constants.

For discharge-based extreme flow factor methods the generic equation is:

$$EFF = c_1 + \frac{c_2 + (m_1 Q)^{e_1}}{c_3 + (m_2 Q)^{e_2}}$$

where Q is total sanitary (base) load and c1, c2, c3, m1, m2, e1, and e2, are constants.

The Extreme Flow Factor Equation Method is defined by the following:

- **Label** - The name of the extreme flow factor method as it will appear in choice lists.
- **Function of** - The type of extreme flow factor method, population-based or discharge-based, associated with the equation. The extreme flow setup function is selected when a new extreme flow factor method is created.
- **Unit** - The unit in which the formula is defined. The coefficients in the equation are dependant on this unit.
- **Cutoff Value** - The maximum possible extreme flow factor for peaking methods. This is used to prevent unrealistically high values for small populations or land areas.
- **Equation coefficients** - *c1, c2, c3, m1, m2, e1,* and *e2*

**You can use the Plot option to see the extreme flow factor method range. Also, use the Notes tab to enter any information relevant to the equation.**

### 12.5.2 Extreme Flow Factor Table Properties

The **Extreme Flow Factor Table Method** is defined by:

- **Label** - The name of the extreme flow factor method as it will appear in choice lists.
- **Function of** - Type of the extreme flow factor method, population-based or discharge-based, associated with the equation. The extreme flow setup function is selected when a new extreme flow factor is created.
- **Extreme Flow Factor Table** - EFF vs. base load (or total dry weather discharge) values.

**Extreme flow factor values that fall outside of the range of boundary values will be assigned the closest in-range value, either the first (lowest) or last (highest) value. Also, use the Notes tab to enter any information relevant to the table.**

## 12.6 Section Size

### 12.6.1 Section Size Library

A listing of typical component sizes for specific material types used in storm and sanitary sewer design is provided. Occasionally, there may be situations where specific component sizes or types will not be physically available for a project. You may wish to remove these component types from the listing so that they are not used during automatic design. Similarly, your project may require a size that is not in the list provided.

This powerful engineering library system allows you to add and remove section sizes, or edit the properties of an existing section size. The default list includes hundreds of predefined section sizes in five different shapes. Each section size can be made available for any of the defined materials.

The five section shapes are:

- Arch
- Box
- Circular

- Horizontal Ellipse
- Vertical Ellipse

Section size properties are edited through the **Engineering Library Manager** and the **Engineering Library Editor**.

To access the **Section Size Properties** dialog, select **Tools / Engineering Libraries** from the pull-down menus. Click the **Edit** column of the **Section Size Library**, and click the button that appears. Select a section size from the list, and click the **Edit** button.

### 12.6.2 Arch Section Size Properties

The **Arch Section Size Properties** tab in the **Section Size Properties** dialog is used to edit the physical properties for arch sections. These properties control how the section is displayed, and define the hydraulic parameters used during calculations.

- **US Label** - Enter the label for the section that will be used when the predominant unit system is US Customary.
- **SI Label** - Enter the label for the section that will be used when the predominant unit system is System International (SI).
- **Span** - Enter the span (width) of the arch shape.
- **Rise** - Enter the rise (height) of the arch shape.
- **Full Area** - Enter the full cross-sectional area for the arch.
- **Bottom Radius** - Enter the radius for the bottom curve of the arch.
- **Bottom Distance** - Enter the distance from the bottom of the arch section to the pivot point for the top curve.
- **Corner Radius** - Enter the radius for the corner curves of the arch.
- **Top Radius** - Enter the radius for the top curve of the arch.

To access the **Arch Section Size Properties** dialog, select **Tools / Engineering Libraries** from the pull-down menus. Click the **Edit** column for the **Section Size Library**, and click the button that appears. Choose the desired arch size from the list, and click the **Edit** button.

### 12.6.3 Box Section Size Properties

The **Box Section Size Properties** tab in the **Section Size Properties** dialog is used to edit the physical properties for box sections. These properties control how the section is displayed, and define the hydraulic parameters used during the calculation.

- **US Label** - Enter the label for the section that will be used when the predominant unit system is US Customary.
- **SI Label** - Enter the label for the section that will be used when the predominant unit system is System International (SI).
- **Span** - Enter the span (width) of the box shape.
- **Rise** - Enter the rise (height) of the box shape.
- **Full Area** - The calculated full area for the box.

To access the **Box Section Size Properties** dialog, select **Tools / Engineering Libraries** from the pull-down menus. Click the **Edit** column for the **Section Size Library**, and click the button that appears. Choose the desired box size from the list, and click the **Edit** button.

### 12.6.4 Circular Section Size Properties

The **Circular Section Size Properties** dialog is used to edit the physical properties for circular sections. These properties control how the section is displayed, and define the hydraulic parameters used during the calculation.

- **US Label** - Enter the label for the section that will be used when the predominant unit system is US Customary.
- **SI Label** - Enter the label for the section that will be used when the predominant unit system is System International (SI).
- **Diameter** - Enter the diameter of the section.
- **Full Area** - The calculated full cross-sectional area for the circular section.

To access the **Circular Section Size Properties** dialog, select **Tools / Engineering Libraries** from the pull-down menus. Click the **Edit** column for the **Section Size Library**, and click the button that appears. Choose the desired circular size from the list, and click the **Edit** button.

### 12.6.5 Ellipse Section Size Properties

The **Ellipse Section Size Properties** dialog is used to edit the physical properties for both horizontal and vertical ellipse sections. These properties control how the section is displayed, and define the hydraulic parameters used during the calculation.

- **US Label** - Enter the label for the section that will be used when the predominant unit system is US Customary.
- **SI Label** - Enter the label for the section that will be used when the predominant unit system is System International (SI).
- **Span** - Enter the span (width) of the ellipse shape.
- **Rise** - Enter the rise (height) of the ellipse shape.
- **Full Area** - Enter the full cross-sectional area for the ellipse.
- **a0, a1, a2, a3, a4** - Enter the five coefficients to the $4^{th}$ order polynomial equation, which describes the relationship between the wetted perimeter (in inches) of the ellipse and the depth/rise ratio, as follows:

$$p = a_0 + a_1\left({}^{d}\!/_{r}\right) + a_2\left({}^{d}\!/_{r}\right)^2 + a_3\left({}^{d}\!/_{r}\right)^3 + a_4\left({}^{d}\!/_{r}\right)^4$$

| | | | |
|---|---|---|---|
| Where: | p | = | Wetted Perimeter (in) |
| | d | = | Water depth in the pipe (ft) |
| | r | = | Pipe rise (ft) |

- **Equivalent Diameter** - Enter the diameter of a circular section that is equivalent to the ellipse.
- **Flow Area Factor** - Enter the flow area factor for the ellipse. This is the ratio of the calculated full area (π * rise * span / 4) to the specified full area.

To access the **Ellipse Section Size Properties** dialog, select **Tools / Engineering Libraries** from the pull-down menus. Click the **Edit** column for the **Section Size Library**, and click the button that appears. Choose the desired ellipse size from the list, and click the **Edit** button.

### 12.6.6 Available in Materials

Each section size is available in a variety of materials, which means you only have to define the physical characteristics of the section once.

This tab in the **Section Size Properties** dialog is used to specify the materials in which the section size is available. Just add those materials to the table from the list provided. The list of materials, also accessible in the **Material Library**, can also be edited. See the **Engineering Library Editor** for more details.

Section size materials are edited through the engineering library system by using the **Engineering Library Manager** and the **Engineering Library Editor**.

# 12.7 Material Properties

## 12.7.1 Material Library

A customizable library of materials is provided. Pipes are constructed from various materials. It is often useful to specify the material of the pipes and channels/ditches in your hydraulic and hydrologic models. Materials provide the pipe or channel with a default value for the roughness coefficient used in the friction equations. Therefore, a material must be defined with the following properties:

- **Label** - Name of the material as it will appear in material selection lists.
- **Culvert Inlet Material Type** - Limits the type of culvert inlets that are available when the material is used as the culvert material (used in CulvertMaster). The inclusion of this property allows the sharing of libraries among Haestad Methods' products.
- **Manning's Coefficient** - Default value for Manning's n. This is a number generally between 0.009 and 0.300.
- **Roughness Height** - Default value for absolute roughness height. This will be used in conjunction with the Darcy-Weisbach friction equation. The roughness height has units of length, typically mm or ft.
- **Kutter's n Coefficient (StormCAD and SewerCAD)** - Default value for Kutter’s formula. This is a unitless number generally between 0.009 and 0.300.
- **C Coefficient** - Default value for Hazen William's C. This is a unitless number generally between 60 and 150.

The check boxes next to each item specify whether the friction method will be available for the material. For example, some materials, such as asphalt, only have Manning’s n values defined.

# 12.8 Minor Loss

## 12.8.1 Minor Loss Properties

An editable library of minor losses is provided. Minor losses are used on pressure pipes and valves to model headlosses due to pipe fittings or obstructions to the flow. A minor loss is defined with the following properties:

- **Label** - Name of the minor loss as it will appear in choice lists.
- **Type** - General type of fitting or loss element. This field is used to limit the number of minor loss elements available in choice lists. For example, the minor loss choice list on the valve dialog only includes minor losses of type valve. You cannot add or delete types.
- **K Coefficient** - Headloss coefficient for the minor loss. This unitless number represents the ratio of the headloss across the minor loss element to the velocity head of the flow through the element.

# 12.9 Unit Sanitary (Dry Weather) Load

SewerCAD defines unit sanitary loads in editable **Engineering Libraries**, allowing you to edit predefined unit sanitary loads and insert new ones. A unit sanitary load is used to specify loads to a sewer system for a user-selected loading unit. Unit sanitary loads can be either population-based or non-population-based. Population-based unit sanitary loads specify a load to the sewer system as a function of the contributing population. Non-population-based loads specify loads based on service area, discharge, or user-defined counts.

**Throughout the program and documentation, the term Unit Sanitary (Dry Weather) Load(s) may be abbreviated as Unit Dry Load(s).**

## 12.9.1 Population-Based Unit Sanitary (Dry Weather) Load Properties

The most common way of specifying sanitary loads to a sewer system is to make them proportional to the contributing population. Population-based unit sanitary loads define loads as a function of adjusted contributing population. You can select the population loading units that will be used and the unit load per population unit. For example, the unit sanitary load, Home (Average), specifies Resident as the population loading unit, and 280 l/d per Resident as the unit load per population unit.

The population-based unit sanitary load is defined by:

**Label** - The name of the unit sanitary load as it will appear in choice lists.

**Loading Unit Type** - Type of the unit sanitary load: population-based, area-based, discharge-based, or used defined count-based. **Loading Unit Type** is selected upon creation of a new unit sanitary load, and therefore this field is not editable.

**Loading Unit** - The base unit used to define the unit load. **Loading Unit Type** defines the list of available loading units, such as resident, capita, guest, or employee.

**Unit Load** - The amount of flow contributed per loading unit.

**Population-based sanitary loads can be peaked using any Extreme Flow Factor Method. Also, use the Notes tab to enter any information relevant to the unit sanitary load.**

## 12.9.2 Non-Population-Based Unit Sanitary (Dry Weather) Loads

Non-population-based unit sanitary loads can be area-based (function of contributing area), discharge-based (function of direct discharge), or count-based (function of a user-defined count). In addition to loading units and unit load per specified loading unit, non-population-based unit sanitary loads are defined by the following properties:

**Label** - The name of the unit sanitary load as it will appear in choice lists.

**Loading Unit Type** - Type of the unit sanitary load: population-based, area-based, discharge-based, or user-defined count-based. **Loading Unit Type** is selected upon creation of a new unit sanitary load.

**Loading Unit** - The base unit used to define unit load. **Loading Unit Type** defines the list of available loading units.

**Unit Load** - The amount of flow contributed per loading unit.

**Population Equivalent** - Count of adjusted population per loading unit. Adjusted population is used with population-based extreme flow factor methods. For area based loads, this is essentially a population density, or population per unit area.

**Report Adjusted Population** - If this option is toggled ON, the adjusted population will be reported with other populations. If the option is OFF, adjusted population will be used only to calculate Extreme Flow Factors and will not be reported as part of the total population.

**Non-population-based sanitary loads are typically peaked using a discharge-based extreme flow factor method. If you wish to use a Population-Based Extreme Flow Factor Method with a non-population-based sanitary load, the Population Equivalent must be specified.**

### 12.9.3 Area-Based Unit Sanitary (Dry Weather) Loads

Area-based unit sanitary loads are commonly used to specify industrial loads and steady inflows. Use these unit sanitary loads whenever your load is specified as a function of contributing area.

### 12.9.4 Discharge-Based Unit Sanitary (Dry Weather) Loads

Discharge-based unit sanitary loads are used to directly specify loads without specifying them on the basis of some other count, such as population or area.

There are two general ways to use discharge-based loads:

- Specify 1.0 discharge unit (e.g. l/day, gpd, cfs, etc.) as the unit load. Then, when using the load, specify the total desired load for the loading unit count. For example, you can create a load called Liter per Day whose loading unit type is Discharge, loading unit is l/day, and unit load is 1.0. When you use this load at a manhole, a wet well, or a pressure junction, you specify 50.0 as the loading unit count. This yields a base load of 50 l/day.
- Specify total desired load as the unit load. Then, when using the load, only specify 1.0 as the loading unit count. For example, you can create a load called Industry XYZ whose loading unit type is Discharge, loading unit is l/day, and unit load is 2000.0. When you use this load at the manhole, wet well, or pressure junction, you would specify 1.0 as the loading unit count. This yields a base load of 2000 l/day.

In other words, you can specify a unit load of 1.0 in the **Unit Sanitary Load Library** and determine the total load at each node through the loading unit count, or you can specify the total load in the **Unit Sanitary Load Library** and then have a loading unit count of 1.0.

### 12.9.5 Count-Based Unit Sanitary (Dry Weather) Loads

Count-based unit sanitary loads should be used for any load that is not area, population, nor discharge-based. These loads allow you to specify any loading unit such as loading per vehicle, machine, or anything else.

**Loading units in user-defined counts are treated only as labels. Conversion between these units is always 1 to 1.**

Notes

# Chapter 13

# GIS and Database Connections

## 13.1 Overview

Haestad Methods' GIS/Database Connection feature provides the modeler with the ability to dynamically exchange data with a variety of applications. You can establish a "connection" between your hydraulic model and relational and non-relational database management systems (RDBMS and DBMS), spreadsheets, and ESRI Shapefiles. Throughout the rest of this chapter, the term "external file" will be used to generically refer to any one of these types of files. Where information pertains to a specific type of external file, that type will be used.

The GIS/Database Connection system is extremely powerful. It can be used to update hundreds or thousands of database records with a few clicks of the mouse. This chapter provides detailed information on the structure and behavior of the system so that it can be used more effectively.

The purpose of the GIS/Database Connection system is to provide you with a safe and convenient means of exchanging data with external files. This system has several advantages over simply providing an open file format for direct manipulation by the end user.

**Generality** - Open file formats have a specific form that must be adhered to. This restrictiveness is problematic for both the developer and the end user. Developers are now under additional constraints when modifying the software. They must be cognizant of the fact that users may depend on this format, and are therefore less free to modify it. The end user, on the other hand, has no control over this format, and is at the mercy of the developer. A new version may change the format completely, and all of your existing data must be converted. In addition, the file format is rarely convenient for an end user since it is typically chosen for efficient processing by the program. The GIS/Database Connection system allows you to exchange data between the model and any arbitrarily defined external files. This flexibility allows you to set up a database or spreadsheet, and it frees the developer to use a file format that is most efficient for the program.

**Data Protection** - Open file formats can typically be modified by anyone, often without the knowledge of the modeler. By providing an interface to exchange data, the model is protected from inadvertent changes. The modeler is in complete control of when and how the model or external files are updated.

**Type Coercion** - Quite often the external files do not store the data using the format expected by the hydraulic model. For example, a database may store the length of a pipe using single precision floating point numbers, whereas the model works with double precision floating point numbers. When exchanging data between the model and the external file using the GIS/Database Connection system, the data is coerced from one type to the other automatically.

**Unit Conversion** -The quantities used in hydraulic models almost always have some unit associated with them. For example, pipe lengths are typically expressed in meters or feet. General purpose database and spreadsheet applications do not support the concept of unitized numbers. A pipe length, for example, is simply represented as 100.0. Is that 100.0 meters or 100.0 feet? The GIS/Database Connection interface allows you to specify the database unit so the numbers can be converted from the model unit to the database unit and vice versa.

Virtually all model inputs and calculated results can be exchanged through the GIS/Database Connection system. The system not only supports the update of existing model elements and external file records, but also the creation and deletion of these elements and records. For example, by performing a **Sync In** operation (explained in detail below), an entire hydraulic model can be built from data stored in a spreadsheet. Likewise, an empty spreadsheet can be completely populated with data from an existing hydraulic model by performing a **Sync Out** operation. The spreadsheet can be kept synchronized with the hydraulic model over the course of a project as new elements are added or deleted, and the input and output data is modified.

The GIS/Database Connection system has a three-tiered architecture:

- Connections
- Table or Shapefile Links
- Field Links

The first tier is the Connection. Connections are organized and managed by **Connection Managers**. There are two types of **Connection Managers**: a **Database Connection Manager** and a **Shapefile Connection Manager**. As the names imply, the first manages connections to databases and spreadsheets, and the second manages connections to ESRI Shapefiles. The **Connection Managers** are similar, and provide an interface for adding, editing, deleting, duplicating, and synchronizing Connections.

To exchange data between the model and external files, a Connection must be created and then synchronized. The two synchronization operations that can be performed on a Connection are **Sync In** and **Sync Out**. **Sync In** synchronizes the model to the data contained in external files. In this case, the model acts as a "consumer" of the data, and external files act as the data "provider". **Sync Out** synchronizes external files to the data contained in the model. Thus, for **Sync Out**, the model is the data provider and external files are the consumers. Exactly what data is exchanged during synchronization depends on how the Connection is defined. Intuitively, a Connection must specify which files are to be connected to the model, and what data in each file is to be exchanged.

The second tier is the Table or Shapefile Link. A Database Connection uses these links to gather and store information. Each Connection can contain one or more Table or Shapefile Links. Each of these links specifies the type of external file with which to exchange data (implied with Shapefile links), the name of the file, and, if the file contains multiple tables, which table within the file is of interest.

The third tier of the system is the Field Link. Each Table or Shapefile Link uses one or more Field Links to specify exactly what data in the external file is going to be exchanged. A Field Link defines the fundamental mapping between a field in an external file and a field in the model. For example, a field link may be used to "map" the GRND_FT field of an external database file to the Ground Elevation attribute of the model.

In summary, a Connection defines a link between the model and external files. Table or Shapefile Links and Field Links are used to specify files, tables, and fields to be linked. Once a Connection is created, it can be synchronized in or out. The synchronization action will update models ("in" direction) or the external files ("out" direction).

The rest of this chapter provides details on the dialogs and windows used to interact with the GIS/Database Connection system. Although Database Connections and Shapefile Connections are similar in concept, there are differences in the interfaces and options. Therefore, they will be discussed in separate sections.

# 13.2 Database Connections

## 13.2.1 Database Connection Manager

This manager, accessed by selecting **File / Synchronize / Database Connections** from the pull-down menu, helps you track and work with database connections. On the left side of this dialog is a list of the current database connections.

There are several options available in the **Database Connection Manager**, including:

- **Add** - Creates a new database connection using the **Database Connection Editor**.
- **Edit** - Changes the configuration of the currently selected connection. This will open the **Database Connection Editor**, where you can rename the connection, change the associated database files, and perform other changes to the connection configuration.
- **Duplicate** - Creates a connection identical to the selected one. This feature is very helpful when defining two or more connections with many similar attributes.
- **Delete** - Removes the selected connection from the list.
- **Synchronize In** - Updates the network attributes from the databases defined in the selected connection.
- **Synchronize Out** - Updates all databases in the connection from the current status of the model.
- **Reset** - Returns the highlighted standard database import or export connection to default settings.

When synchronizing in, output fields such as hydraulic grade line or computed pipe flow will not be updated. If an attempt is made to update an output field during a **Synchronize In** operation, a "Read Only Warning" will be issued in the status log, indicating which attribute could not be updated.

When synchronizing out, all mapped information will be overwritten in the database files, including input and output conditions.

**If you do not want your input values overwritten upon synchronizing out, simply duplicate the connection. Then, edit one connection such that it includes only the values you want to synchronize in, and one that includes only the values you want to synchronize out.**

**When synchronizing out, be sure that the model element labels are of the same data type as the database column to which you are mapping. Otherwise, synchronizing out to the database will yield erroneous results. For example, if you were to synchronize in from a database where your pipe identifier was numeric, then any changes or additions to the pipes in the model should also use a numeric-labeling scheme. To assure the consistency in type in this case, select Element Labeling from the Tools menu and remove the appropriate element prefixes before any changes are made to the model.**

To access the **Database Connection Manager**, select **File / Synchronize / Database Connections**. From this dialog, there are two ways to get to the **Database Connection Editor**. You can click **Add** to create a new connection, or select **Edit** to change an existing connection.

## 13.2.2 Standard Database Import/Export

The **Database Connection Manager** is initialized with four standard database connections for importing and exporting model data using simple **File** menu commands. These standard connections are as follows:

- **[Project Export - SI]** - Used for the **File / Export / Database** command when the global unit system is set to System International.

- **[Project Export - US]** - Used for the **File / Export / Database** command when the global unit system is set to US Customary.
- **[Project Import - SI]** - Used for the **File / Import / Database** command when the global unit system is set to System International.
- **[Project Import - US]** - Used for the **File / Import / Database** command when the global unit system is set to US Customary.

The purpose of the standard database connections is to provide a powerful yet easy-to-use method of exposing the model data to external applications using a standard database format, Microsoft Access database (.mdb). This method is powerful because it provides you with all the flexibility and functionality of a user-defined database connection, such as unit conversion and type coercion. It is easy to use because it is predefined with all of the standard model data, and requires nothing more than a file name to execute.

The standard database connections are almost identical to user-defined database connections with the following exceptions:

- Standard connections cannot be deleted.
- The label of a standard database connection cannot be changed.
- The target database for a standard database connection is determined at the time it is synchronized. During a **Synchronize In** operation, you will be prompted to choose an existing Microsoft Access Database (.mdb). During a **Synchronize Out,** you will be prompted for the name of a new Access database. If an existing filename is chosen, a warning will indicate that the existing file will be overwritten.
- The field names of the external database tables are editable from within the **Table Link Editor**.
- The **Database Type** on the **Table Link Editor** cannot be changed.
- Standard connections can be reset to their factory default values. To do this, select a standard connection from the list in the **Database Connection Manager**, and click the **Reset** button.

By default, the standard database connections include a table link for each element type, and field links for all the attributes related to that element type, with some minor exceptions. The default units for the specified unit system (SI or US) are used for unitized attributes. The **Key Label** field is designated as the key field for each of the table links, and it is created as an index for the table during database creation. No duplicates are allowed.

As noted above, the field links external field names can be edited directly within the **Table Link Editor**. It is valid to have more than one internal attribute "mapped" to a single external field name. Although this is not the case for the standard connections in their factory default state, you can create this condition. Under this condition, the following behaviors will be observed:

- **Import (Synchronize In)** - All of the attributes will be populated with the value of the database field if it is a valid value for the specified attributes.
- **Export (Synchronize Out)** - The database field will be populated with the last non-blank attribute value.

**If an existing filename is chosen during export, the existing database file will be overwritten. Therefore, any custom tables, queries, or forms present in that database will be lost.**

**Model data that are typically a collection of data (e.g. SewerCAD unit sanitary loads, StormCAD watershed areas and rational C coefficients, and WaterCAD junction demands) cannot be written to a single record, and are therefore not exported to the database. However, if these collections only contain a single item, that single item will be transferred to and from the database during export and import.**

By default, the Standard Database Export creates Microsoft Office 2000 Access files. These files cannot be read with Office 97. If you want to use Office 97, you need to use a text editor to edit the HAESTAD.INI file located in your HAESTAD directory, and replace the line:

ConnectionDatabaseFormat=0

with:

ConnectionDatabaseFormat=3

Basically, a value of 3 results in the program creating an Office 97 Access file, whereas a value of 0 will have the program generate an Office 2000 Access file.

Use the **File / Import / Database** menu item to import data using the standard database connections.

Use the **File / Export / Database** menu item to export data using the standard database connections.

Use the **File / Synchronize / Database Connections** menu item to view or edit the standard database connections.

### 13.2.3 Database Connection Editor

The **Database Connection Editor** is used for defining the group of table links to be included in the connection. The **Database Connection Editor** has tabs for **Database Connection** and **Synchronization Options.**

There are three standard operation buttons at the bottom of the dialog:

- **OK** - Accepts the current condition of the connection, including any changes that have been made.
- **Cancel** - Closes the **Database Connection Editor** without saving any changes.
- **Help** - Opens the context-sensitive Help system.

To access the **Database Connection Editor**, select **File / Synchronize / Database Connections** from the pull-down menu. This will open the **Database Connection Manager**. From this dialog, there are two ways to get to the **Database Connection Editor**. You can click **Add** to create a new connection, or select **Edit** to change an existing connection.

#### Database Connection Tab

The **Database Connection** tab of the **Database Connection Editor** provides an interface for the standard attributes of a connection. It contains the following:

- **Connection Label** - A required unique alphanumeric identification for the connection. This is the label that appears in the list on the **Database Connection Manager** dialog.
- **Table Links** - Provide basic information about each table link, such as the referenced database file, the specific table within the database, and the type of table that is referenced. A table link can be highlighted from the list, at which point the following commands can be performed using the buttons on the right side of the dialog:
  - **Add** - Adds a new table link. If there are no table links currently defined for this connection, this will be the only button available.
  - **Edit** - Changes the characteristics of the selected table link, such as the referenced file or table, or the mapping of the table's field links.
  - **Duplicate** - Duplicates the selected table. This command is very helpful when defining two or more table links with similar attributes.
  - **Delete** - Deletes the selected table link from the connection.

## Synchronization Options Tab

The **Synchronization Options** tab of the **Database Connection Editor** provides an interface for some of the behaviors of the connection. These options cannot be accessed until the Table Links are defined, and are as follows:

- **Add objects to destination if present in source** - If this option is selected, when performing a **Synchronize Out** for example, elements that are present in the model but are not found in the database file will be created in the database. If this is not checked, only the elements that are present in both the model and the database will be updated.
- **Prompt before adding object** - If this is checked, you will get a dialog notifying you of each unmapped element in the source, and asking if you would like to create a new element in the destination. If this is not checked, the additional elements will be automatically created in the database.
- **Remove objects from destination if missing from source** - If this is checked when synchronizing out, elements that are present in the database but not in the model will be deleted from the database. If this is not checked, the unmapped elements will be ignored.
- **Prompt before remove** - When this box is checked a dialog will appear notifying you of each unmapped element in the destination and asking if you would like to remove that element. If the box is not checked, the additional elements will be automatically removed from the database.

> **In order to be successfully created from the database, pipe elements must have a Start and Stop node associated with them. This association can be established by mapping the '+ Start Node' and '+ Stop Node' attributes in the pipe table link, or by the '+ In Link' or '+ Out Link' of a node table link. Mapping both the pipe table and node table attributes may result in the reading of redundant data causing the connection to fail.**
>
> **By default, elements created from a database are located at coordinate (0,0). This behavior can be overridden by mapping the X and Y or Northing and Easting attributes of the node elements.**

## Database Table Link Editor

The **Table Link Editor** is a tool for defining or modifying a table link. This dialog is separated into two groups, one dealing with the file and table information, and the other dealing with the field links (attribute mapping).

The general table link information includes:

- **Database Type** - Type of database to which the link will be made. There are many types of external files that can be linked into the model. Among these are Btrieve, Dbase, Excel, FoxPro, Jet (.mdb files, such as Access), Lotus, and Paradox, as well as Oracle, Sybase, SQL Server, or any other Open Database Connectivity (ODBC) compliant database.
- **Database File** - File referenced by the table link. To browse directories and specify a file path, click the **ellipsis (...)** button.
- **Database Table** - Once the external file has been selected, it will be scanned for tables (or worksheets), which will then be available for selection from this field. Only one table can be linked for each table link, but table links can be easily duplicated and edited from the **Database Connection Editor**.
- **Table Type** - Defines the type of data that can be mapped for this particular table link. For example, a Pipe type of table link means that the available model attributes to be mapped are items such as material, roughness coefficient, flow rate, and velocity.
- **Key\Label Field** - Key by which the entire database-model mapping is defined. The model references each element by a unique alphanumeric label, and the database must contain the same labels in one of

the columns. If the key field for you data type is numeric, you will want to be sure that your model labels include numbers only. Make sure that there are no duplicate element labels/keys within the data source.

The **Field Links** group is a manager for the attribute mapping. The tabular list in this group has three field columns:

- **Model (SewerCAD, StormCAD, WaterCAD)** - Each item in this column is an attribute in the model that is being mapped to the database. The list of available attributes depends on the type of the table.

**Clicking the button in the Field Links cell will open the Quick Attribute Selector. This will allow you select attributes from organized categories to more easily find needed attributes.**

- **Database** - Each item in this column is a heading from the database table, which correlates to the item in the model being mapped.
- **Unit** - This column defines the units of the values in the database. During a synchronization operation, the values will automatically be converted to the appropriate units to maintain the desired unit systems in both the model and the database. No conversion on your part is required.

In addition to the standard table operations of **Insert**, **Duplicate**, and **Delete**, the **Field Links Manager** offers the following additional operation:

- **Select** - Opens the **Select Field Links** dialog for an efficient method of selecting the fields of interest from the available model fields.

To access the **Table Links Editor**, select **File \ Synchronize \ Database Connections** from the pull-down menu. This will open the **Database Connection Manager**. Click **Add** to create a new connection, or select **Edit** to change an existing connection. From the **Database Connection Tab** of the **Database Connection Editor**, click **Add** or **Edit**.

## Select Field Links

The **Select Field Links** dialog provides an easy-to-use interface for populating the **Field Links** group of the **Table Link Editor** or **Shapefile Link Editor**.

The dialog contains two lists:

- **Available Items** - Model attributes that are available for mapping in the current Table or Shapefile Link.
- **Selected Items** - Model attributes that have been selected for mapping.

The following buttons are provided to move items from one list to the other:

- **[>]** - Moves the selected item or items from the **Available Items** list to the **Selected Items** list.
- **[>>]** - Moves all items from the **Available Items** list to the **Selected Items** list.
- **[<]** - Moves the selected item or items from the **Selected Items** list to the **Available Items** list.
- **[<<]** - Moves all items from the **Items Selected** list to the **Available Items** list.

**The Select Field Links dialog provides functions similar to the Table Setup dialog. Please refer to Selected Table Columns help for information on topics such as selecting multiple attributes.**

## 13.2.4 ODBC

### About ODBC

ODBC, which stands for Open Database Connectivity, is a standard programming interface developed by Microsoft for accessing data in relational and non-relational database management systems (DBMS). Using ODBC, applications such as Haestad Methods' engineering software can access data stored in many different PC, minicomputer, and mainframe DBMS, even though each uses a different storage format and programming interface.

The ODBC architecture conceptually consists of three parts:

1. **The application program** - The Haestad Methods product.
2. **The Data Source Administrator Program** - Embedded in Microsoft Windows.
3. **The low-level drivers for accessing specific databases** - Supplied by your database vendor.

Although most computers with Windows will have ODBC present, the exact databases you can interface via ODBC will depend on the databases and drivers installed on your computer.

ODBC is powerful because it is generic and can access many database systems, including mainframe, GIS, and legacy systems. However, because ODBC must be general, it is slower, more complex, and more difficult to use than working directly with a database. When you have the option to work directly with a database, you will usually find it faster and easier than going through ODBC.

For specific information about ODBC in your environment, see your database vendor's documentation. For general information on ODBC, see the online Help for the ODBC Data Source Administrator Program. To find the Administrator Program, go to the Control Panel of your computer and double-click the **ODBC** icon. Choose the **Help** button on the dialog that appears, and go to the Help Contents.

### ODBC Database Type

The first field of the database connection **Table Link Editor** is the **Database Type**. The list box displays the external databases and versions supported by the Database Connection feature. One of the Database Types you can select is ODBC. This does not refer to a specific database or version. It is actually a link to the ODBC Data Source Administrator Program running on your computer. This link will provide an interface between the Haestad Methods' Database Connection and a specific DBMS and source database file.

### ODBC Database File

If you have selected ODBC as the **Database Type**, when you click the **ellipsis (...)** button next to the **Database File** field the ODBC Data Source Administrator Program will take over and offer a list of the ODBC data sources installed on your computer. Depending on how your computer is configured, you may see database systems or actual database files from which to choose.

**You will also see database systems such as Microsoft Jet or Excel that are supported directly via choices in the Database Type list. In general, the Database Connection feature will work faster by choosing these database systems directly rather than going through ODBC.**

If you choose a data source from the Administrator Program, upon returning to the **Table Link Editor** you will see an ODBC "connect string" in the **Database File** field, rather than a file path. This connect string is a series of key = value pairs, separated by semicolons. It specifies the database location, security parameters, and access options needed by the particular ODBC driver you are using. In general, you should not edit this string in any way as you could introduce an error that would prevent the ODBC driver from accessing the data source you have selected.

**If you are unable to successfully synchronize to the data source using the default form of the ODBC string, it is possible that you may need to add some parameters to the string that are specific to your environment. See your database vendor's ODBC documentation for details.**

#### Synchronizing Via ODBC

Once you have successfully created and entered the data for a database connection that uses ODBC, the **Synchronize In** and **Synchronize Out** operations perform as they do for any other database format. However, ODBC databases are accessed with slightly different internal mechanisms, and thus may generate different error conditions. If a synchronization fails to complete, see the status log for error messages. Note the project or database object the program was processing when the error occurred. Refer to your database vendor's documentation for detailed information on any errors reported.

Using ODBC to access SQL Server databases will result in an error #3197 if the synchronization attempts to delete a database record. To avoid this error uncheck **Remove Objects** on the **Synchronization Options** tab of the **Database Connection Editor**.

#### ODBC Database Tables and Fields

There are many complexities in successfully accessing ODBC databases. You will know if there are problems on your machine because the Database Table or other database-related fields will not have any entries in the associated drop-down lists.

If this happens, confirm that ODBC is installed and operating correctly on your computer. Double-check that the ODBC data source you are trying to reference actually exists and is accessible by other programs in your environment. Check the HAESTAD.LOG file for error messages pertaining to ODBC. If none of these steps helps you correct the problem, please call Haestad Methods' Technical Support.

Given the diversity of ODBC database drivers and the difficulty of reproducing your networked computing environment, we cannot guarantee that the Database Connection feature will function with all ODBC databases. However, we will try to determine the source of your problem and offer a fix or workaround if possible.

If you edit the connect string manually, you will need to re-enter the dependent fields such as **Database Table** and **Field Links**.

### 13.2.5 Sharing Database Connections between Projects

#### Sharing Database Connections between Some Projects, but Not All Projects

When SewerCAD works with database connections, it is using a file with an ".HDC" extension, which stores the information regarding database files, table links, and field mapping.

Upon opening a SewerCAD project file (.SWR), SewerCAD first looks for a file in the same directory and with the same filename as the ".SWR" file but with the ".HDC" extension. If it finds this file, it uses the database connectivity information contained therein. If it does not find this file, then it defaults to a file in the installed SewerCAD directory called "SWRC.HDC".

##### *Sharing Database Connections between Projects*

If you are working on a local drive, and you have several project files all of which reference common connection information, let your project files automatically default to the "SWRC.HDC" file. Any connectivity changes that you work on in one project will be automatically reflected when you open any other project.

If there are several people working on different projects on different computers but still wishing to have common connectivity information, the appropriate ".HDC" file can be copied (and renamed if necessary) to the individuals' local drives.

#### *Preventing Database Connectivity Sharing between Projects*

There are times when shared connectivity can be more cumbersome than helpful (such as when there are a lot of projects, each with different database connectivity). At these times, it is more useful to have the connectivity associated with one specific project, rather than with all projects. To do this, simply copy the "SWRC.HDC" file from the installed SewerCAD directory to the same location as your project file, and rename "SWRC.HDC" to have the same name as your ".SWR" file.

For example, if your SewerCAD project file is PROJECT1.SWR, rename SWRC.HDC to PROJECT1.HDC. The connections in PROJECT1.SWR can then be modified without the effects being reflected in any other projects, and without seeing the effects of changes made to other projects.

### 13.2.6 Database Connection Example

To connect your model to an external file, take the following steps:

From the File menu, select Synchronize / Database Connections to open the Database Connection Manager. Click Add.

1. In the **Database Connection Editor**, type a label for your Connection.
2. Click **Add** to create a new table link. This will take you to the **Table Link Editor**.
3. Select the type of file to which you would like to link, and then click the **ellipsis (…)** button to browse for and select your database file.
4. Choose the table to which you would like to link, and the type of table.
5. Choose the **Key/Label Field** to define the column in the database that contains the labels of the elements to be synchronized.
6. Define as many field links as you want by selecting the model attribute and the associated database column and unit.
7. Click **OK** to exit the **Table Link Editor**.
8. Click OK again to exit the Database Connection Editor.
9. You should be back at the **Database Connection Manager**. You can leave this dialog and return to the model, or you can choose to **Synchronize In** to the model from the database, or **Synchronize Out** to the database from the model. Click **OK** to save changes and exit back to the model. Click **Cancel** not to save changes and exit back to the model.

## 13.3 Shapefile Connections

### 13.3.1 Shapefile Connection Manager

This manager is identical to the **Database Connection Manager**, except that it helps you to track and work with Shapefile connections rather than database connections. Only a brief description of each dialog control is presented here. Please refer to the **Database Connection Manager** topic for a more detailed explanation.

- **Add** - Creates a new Shapefile connection. This will open the **Shapefile Connection Wizard**.
- **Edit** - Changes the configuration of the currently selected connection. This will open the **Shapefile Connection Editor**.
- **Duplicate** - Duplicates the selected connection.
- **Delete** - Deletes the selected connection from the list.

- **Synchronize In** - Updates the network attributes from the Shapefiles linked to the selected connection.
- **Synchronize Out** - Updates all Shapefiles within the connection from the current status of the model.

> **See the Overview at the beginning of this chapter for a general discussion of Shapefile Connections.**

To open the **Shapefile Connection Manager**, select the **File \ Synchronize \ Shapefile Connections** menu item. If there are no existing **Shapefile Connections** you will be prompted to create a new Shapefile. Click **No** if you wish to continue on to the **Shapefile Connection Manager**.

### Shapefile Connection Wizard

The **Shapefile Connection Wizard** provides an easy-to-use interface for defining a new Shapefile Connection. It is similar to the **Shapefile Import Wizard**, but has a few additional steps. The major steps in the wizard are as follows:

- **Label** - Enters an alphanumeric label to uniquely identify the Shapefile Connection.
- **Select Element Types** - Chooses the types of network elements you wish to connect to Shapefiles.
- **Shapefile Synchronization Options** - Specifies the spatial data unit, and configures other options.
- **Import Shapefile Link Editor** - Chooses the Shapefile to which you want to connect and specifies the details of the link.
- **Synchronize Now** - Choose whether you want to synchronize the Shapefile Connection when finished with the wizard. You can choose to synchronize in either direction.

#### *Shapefile Connection Label*

The **Shapefile Connection Label** window allows you to enter a unique alphanumeric label for your Shapefile Connection. This window is presented in the **Import** and **Export Shapefile Connection Wizards**, as well as the **Shapefile Connection Wizard**.

#### *Synchronize Now?*

The last step in the **Shapefile Connection Wizard**, the **Synchronize Now?** Window, allows you to specify whether you wish to synchronize the Shapefile Connection immediately after editing it in the wizard. The following options are available:

- **Synchronize Shapefile Connection** - Check this box if you wish to synchronize the connection immediately upon clicking the **Finished** button. By default this box is checked. If you uncheck it, you will return to the **Shapefile Connection Manager** after clicking the **Finished** button.
- **In** - Select this radio button if you wish to synchronize the connection in to the model. This will update the model data from the Shapefiles linked to the connection.
- **Out** - Select this radio button if you wish to synchronize the connection out to the Shapefiles linked to the connection. This will update the Shapefiles from the model.

## 13.3.2 Shapefile Connection Editor

The **Shapefile Connection Editor** is similar to the **Database Connection Editor**. It offers the tabs for **Shapefile Connection** and **Synchronization Options**.

To use the **Shapefile Connection Editor**, do the following:

1. Select **Synchronize / Shapefile Connections** from the **File** menu.

2. If you do not currently have any Shapefile connections defined, you will be prompted to indicate if you wish to create one now. If you answer **Yes**, you will be automatically taken to the **Shapefile Connection Wizard**.
3. If there are connections already defined, or if you answer **No** to the prompt to create one now, you will be taken to the **Shapefile Connection Manager**. Select **Edit** to open the **Shapefile Connection Editor**.

#### Shapefile Connection

The **Shapefile Connection** tab of the **Shapefile Connection Editor** is similar to the **Database Connection** tab of the **Database Connection Editor**. It contains the following:

- **Connection Label** - A unique alphanumeric identification for the connection. This is the label that appears in the list on the **Shapefile Connection Manager** dialog.
- **Table Links** - List that provides basic information about each Shapefile link, such as the referenced Shapefile, the feature type of the Shapefile, and the type of element which is referenced. As with the other managers, a Shapefile link can be highlighted from the list, at which point the following commands can be performed using the buttons on the right side of the dialog:
  - **Add** - Defines a new Shapefile link. If there are no table links currently defined for this connection, this will be the only button available. Clicking this button invokes the **Shapefile Link Wizard**.
  - **Edit** - Changes the characteristics of the selected Shapefile link, such as the referenced file or the mapping of the Shapefile's field links. Clicking this button also invokes the **Shapefile Link Wizard**.
  - **Duplicate** - Creates an identical Shapefile link to the selected one. This is very helpful when defining two or more Shapefile links with similar attributes.
  - **Delete** - Removes the selected Shapefile link from the connection.

### 13.3.3 Shapefile Link Wizard

The **Shapefile Link Wizard** is used when adding new Shapefile Links to a Shapefile Connection, or when editing the existing links of a Shapefile Connection. The first step of the wizard is bypassed when editing an existing link. The basic steps of the wizard are as follows:

- **Select Element Type** - Similar to the **Select Element Types** window for importing Shapefiles, except that radio buttons are used rather than check boxes. This is because a Shapefile Connection represents a single element type.
- **Import Shapefile** - Choose the Shapefile to which you would like to connect, and the **Key/Label Field** to specify the column in the Shapefile that contains the matching element labels in the network. Define as many field links as necessary. For each link, specify the model attribute, the associated Shapefile column, and the Unit in which the Shapefile attribute is stored.
- **Shapefile Link Summary** - Quick review of the details specified in the wizard.

As with all wizards, you can move forward or backward through the process to make changes. Click the **Finished** button when you are done making changes to the Shapefile Link.

#### Shapefile Link Summary

The **Shapefile Link Summary** window provides an opportunity to review the details of the Shapefile Link before completing the editing process. The following information is provided in the summary window:

- **Type** - Type of element represented by this Shapefile Link.

- **Shapefile** - Full path and file name of the Shapefile referenced by this Shapefile Link.
- **Key/Label Field** - Shapefile field used to map Shapefile records to their corresponding network elements in the model.
- **Attributes Mapped** - Number of Field Links mapped in this Shapefile Link.

### 13.3.4 Import Shapefile Wizard

The **Import Shapefile Wizard** will guide you step-by-step through the process of importing ESRI Shapefiles. These are the basic steps for importing Shapefiles:

- **Select Element Types** - Select the type of network elements you wish to import.
- **Shapefile Synchronization Options** - Specify the spatial data unit and configure other options.
- **Import Shapefile** - Browse to and select the Shapefiles you would like to import, and select the **Key/Label Field** to specify the column in the Shapefile that contains the matching element labels in the network. Define as many field links as necessary. For each link, specify the network attribute, the associated **Shapefile** column, and the **Unit** in which the Shapefile attribute is stored.
- **Create Shapefile Connection** - Select whether you want to establish a Shapefile Connection. The Shapefile Connection allows you to update the Shapefile with values from your model, or to update your model from the Shapefile.

While using the wizard, you can move forward or backward through the process to make changes by clicking the **Next** and **Back** buttons. Click the **Finished** button when you are done making changes to import Shapefiles.

To access the **Import Shapefile Wizard**, select **File / Import / Shapefile** from the pull-down menus.

#### Select Element Types

The **Select Element Types** window is used for selecting the types of network elements that are of interest when importing and exporting Shapefiles, or when creating a Shapefile Connection. The window contains a list of network element types with a check box preceding each type.

To select an element type for Shapefile Import, Export, or Connection, put a check mark in the corresponding box.

#### Shapefile Synchronization Options

Several options are available to customize the Shapefile synchronization process. The **Shapefile Synchronization Options** are available for editing in the **Import Shapefile Wizard** or through the **Shapefile Connection Editor**.

The first group of options is only available when editing a Shapefile Connection. These options are exactly the same as their counterparts in **Database Synchronization Options**, and are as follows:

- Prompt before adding object
- Prompt before removing object

Unlike the **Database Synchronization Options**, the **Shapefile Synchronization Options** do not allow for optionally adding or removing elements. When synchronized, Shapefiles and the model will contain exactly the same number of records for the specified element type. For example, suppose a Shapefile contains a record for the junction labeled J-1. When this Shapefile is synchronized into the model, the model will automatically add a junction labeled J-1 if none currently exists. Likewise, if J-1 is removed from the model and then synchronized out to the Shapefile, the record for J-1 will automatically be removed from the Shapefile. You have no control over this.

The rest of the options are available during the **Shapefile Import Wizard** or through the **Shapefile Connection Editor**.

**Shapefile Unit** - Choose a unit from the available list. This is the unit of the spatial data in the Shapefile. For example, if the X and Y coordinates of the Shapefile represent feet, choose feet from the list. If they represent meters, select meters. This unit must be the same for every Shapefile in the Shapefile Connection. If you wish to import Shapefiles that have different spatial data units, create a separate connection for each unit.

#### *When Missing Connectivity Data*

As noted in the **Table Link Editor** topic, to create a pipe from an external file it is necessary for a pipe to have a start node and stop node associated with it. Typically, these "connectivity" associations are created by synchronizing the '+ Start Node' and '+ Stop Node' attributes of the pipe. Since a Shapefile contains spatial data, it is also possible to establish these associations based on the location of nodes relative to the end points of the pipe. The following options allow you to customize this behavior:

- **Establish By Spatial Data** - Check this box to configure the synchronization so that any missing connectivity data (start node, stop node, or both) for a pipe will be established from the spatial data if possible.
- **Tolerance** - This value represents the distance to be searched when trying to locate nodes for establishing connectivity for a pipe. All nodes within the tolerance of a pipe's end point will be collected, and the closest node will be selected for connection.
- **Create Nodes if None Found** - Check this box if you would like nodes to be created during the synchronization when no nodes are found within the specified tolerance of a pipe's end point. If this box is not checked, and no nodes are found within the tolerance, the pipe will not be created because it has insufficient connectivity data.

### Import Shapefile Link Editor

The **Import Shapefile Link Editor** is similar to the **Database Table Link Editor**. Refer to that topic for detailed information on the following Shapefile Link parameters:

- **Shapefile** - Location of the file that is being referenced by the Shapefile link. This is identical to the Database File parameter of the **Table Link Editor**.
- **Key/Label Field** - Key by which the entire Shapefile/model mapping is defined.
- **Field Links** - Identical to the Field Links group of the Database Table Link Editor.

### Create Shapefile Connection

The **Create Shapefile Connection** window provides an opportunity during a Shapefile Import or Export to specify that a persistent connection containing the Shapefile Links and Synchronization Options be created. This connection can be used at a later time to synchronize the model and the Shapefiles. The **Create Shapefile Connection** window has the following parameters:

- **Add Shapefile Connection** - Check this box if you wish to add a persistent Shapefile Connection to the **Shapefile Connection Manager**. By default, this box is checked.
- **Label** - Specify an alphanumeric label for the connection. This field is only editable when the **Add Shapefile Connection** box is checked.

To access the **Shapefile Import Wizard**, select **File / Import / Shapefile** from the pull-down menus.

### Shapefile Import Example

Follow these steps to import one or more Shapefiles into a new or existing model:

From the File menu, select **Import / Shapefile** to access the **Import Shapefile Wizard**.

1. Choose the element types that you wish to import by selecting one or more of the check boxes in the list, and then click the **Next** button.

2. Configure the options for this import. First, select the unit for the spatial data of the Shapefile. Then, if appropriate for your situation, click the **Establish by Spatial Data** check box in the **When Missing Connectivity Data** group, and enter a value in the **Tolerance** field. For more information regarding these options, refer to the section on **Shapefile Synchronization Options**. Click the **Next** button to proceed to the **Shapefile Link Editors**.

3. You will be presented with an **Import Shapefile Link Editor** for each element type you choose to import. Perform the following steps for each **Import Shapefile Link Editor**:

   a. Enter the name of the Shapefile you wish to import for the specified element type. Click the **ellipsis (…)** button to interactively browse for and select your Shapefile.

   b. Choose the **Key/Label** field to define the column in the Shapefile that maps to the element labels in the model.

   c. Define as many field links as necessary by selecting the model attribute and the associated **Shapefile** column and unit. Use the **Select** button for making the selection process more efficient. Click the **Next** button.

4. Click the **Add Shapefile Connection** check box if you wish to create a persistent link between the Shapefile(s) you are importing and the model. If you choose to create a Shapefile Connection, enter an alphanumeric label to identify the connection. Click the **Finished** button to import the Shapefiles.

### 13.3.5 Export Shapefile Wizard

This program has the capability of exporting network elements in the ESRI Shapefile Format. The ESRI Shapefile is actually three files that together define the spatial and non-spatial attributes of a map feature. In the case of Haestad Methods hydraulic models, map features are network elements (e.g. pipes, junctions). Exporting Shapefiles creates brand new files. If you are exporting a Shapefile to a directory that already contains a Shapefile of the same name, the existing Shapefile will be completely overwritten. If you wish to update the Shapefile rather than overwriting it, use the Shapefile Connection feature.

The major components of the Wizard are as follows:

- **Select Element Types** - Choose the type of network elements you wish to export. Each type of network element will have its own Shapefile associated with it. This component is identical to the Import Wizard's **Select Element Types** component.

- **Export Shapefile Link Editor** - Enter a name for each Shapefile you wish to create. Each Shapefile name must be no more than eight characters in length, and should not be duplicated. Define as many field links as necessary. For each link, specify the network attribute. The Shapefile variable will default to a preset value, which can be edited.

- **Create Shapefile Connection** - Choose whether you want to establish a Shapefile Connection for this Shapefile. The Shapefile Connection allows you to update the Shapefile with values from your model, or to update your model from the Shapefile. This component is identical to the Import Wizard's **Create Shapefile Connection** component.

While using the Wizard, you can move forward or backward through the process by clicking the **Next** and **Back** buttons. When you are finished defining it, click the **Finished** button to create the Shapefile.

To export a specific network element type as a Shapefile, choose **File / Export / Shapefile** from the pull-down menus. This opens the **Export Shapefile Wizard.**

### Export Shapefile Link Editor

The **Export Shapefile Link Editor** is similar to the **Database Table Link Editor**, with the following differences:

- **Shapefile** - The name and location for the file that is being exported. The Shapefile name is limited to eight characters.

The **Field Links** group is used to specify the attributes and Shapefile column headings that you wish to export, as follows:

- **Model** - Each item in this column is an attribute in the model that is being exported to the Shapefile. The list of available attributes depends on the type of table.
- **Shapefile** - Each item in this column is a column heading in the Shapefile being created, which correlates to the item in the model being mapped. By default, the headings are set to an all-capitals abbreviation of the attribute name, with spaces and periods replaced by the underscore character. The column heading can be changed, but must be less than ten characters long and cannot contain periods.

> **The spatial data in the Shapefiles being created will be in the current display unit for map coordinates. For example, if the X and Y or Northing and Easting values in the model are displayed in meters at the time of the export, then the spatial data in the Shapefiles created will also be in meters.**
>
> **The values for the exported attributes will be in the current display units for that attribute. For example, if a junction elevation attribute is displayed in feet at the time of the export, the Shapefile will contain that value in feet.**

### Shapefile Export Example

Follow these steps to export one or more Shapefiles from the model:

From the **File** menu, select **Export / Shapefile** to access the **Export Shapefile Wizard**.

1. Select the element types that you wish to export by selecting one or more of the check boxes in the list, then click the **Next** button.
2. You will be presented with an **Export Shapefile Link Editor** for each element type you choose to export. Perform the following steps for each **Export Shapefile Link Editor**:
   - Enter the name of the Shapefile you wish to create for the specified element type. Click the **ellipsis (...)** button to interactively browse for a directory in which to store the Shapefile.
   - Define as many field links as necessary by selecting the model attribute and providing a name for the associated **Shapefile** column. Use the **Select** button for making the selection process more efficient. Click the **Next** button to continue.
   - Click the **Add Shapefile Connection** check box if you wish to create a persistent link between the Shapefile(s) you are exporting and the model. If you choose to create a Shapefile Connection, enter an alphanumeric label to identify the connection. Click the **Finished** button to export Shapefiles.

## 13.3.6 Sharing Shapefile Connections between Projects

When SewerCAD works with Shapefile connections, it is using a file with an .HSC extension, which stores the information regarding the Shapefiles and field mapping for each element type.

When you open a SewerCAD project file (.SWR), SewerCAD first looks for a file in the same directory and with the same filename but with the .HSC extension. If it finds this file, it uses the Shapefile connectivity information contained therein. If it does not find this file, it defaults to a file in the installed SewerCAD directory called SWRC.HSC.

#### *Sharing Shapefile Connections between Projects*

If you are working on a local drive, and you have several project files that all reference common connection information, let your project files automatically default to the SWRC.HSC file. Any connectivity changes that you work on in one project will be automatically reflected when you open any other project.

If there are several people working on different projects on different computers, but they still wish to have common connectivity information, the appropriate .HSC file can be copied (and renamed if necessary) to the individual local drives.

#### *Preventing Shapefile Connectivity Sharing between Projects*

There are times when shared connectivity can be more cumbersome than helpful such as when there are many projects, each with different Shapefile connectivity. At these times, it is more useful to have the connectivity associated with one specific project, rather than with all projects. To do this, simply copy the SWRC.HSC file from the installed SewerCAD directory to the same location as your project file, and rename it to the same name as your .SWR file.

For example, if your SewerCAD project file is PROJECT1.SWR, rename SWRC.HSC to PROJECT1.HSC. The connections in PROJECT1.SWR can then be modified without the effects being reflected in any other projects.

### 13.3.7 Shapefile Format

An ESRI Shapefile actually consists of three separate files that combine to define the spatial and non-spatial attributes of a map feature. The three required files are as follows:

- **Main File** - A binary file with an extension of .SHP. It contains the spatial attributes associated with the map features. For example, a polyline record contains a series of points, and a point record contains X and Y coordinates.
- **Index File** - A binary file with an extension of .SHX. It contains the byte position of each record in the main file.
- **Database File** - A dBase III file with an extension of .DBF. It contains the non-spatial data associated with the map features.

All three files must have the same file name with the exception of the extension, and be located in the same directory.

### 13.3.8 Shapefile Connection Example

Follow these steps to connect one or more Shapefiles to the model:

1. From the **File** menu, select **Synchronize / Shapefile** Connections.
2. If you do not have any connections currently defined, you will be asked if you want to create a new one now. Select **Yes**. If you already have one or more connections defined, you will go to the **Shapefile Connection Manager**. Click **Add** to access the **Shapefile Connection Wizard**.
3. Provide an alphanumeric label to uniquely identify this new connection. Click the **Next** button.
4. Choose the element types that you wish to import by clicking one or more of the check boxes in the list, and click the **Next** button.
5. Configure the options for this connection. First select the unit for the spatial data of the Shapefile. Then, if appropriate for your situation, click the **Establish by Spatial Data** check box in the **When Missing Connectivity Data** group, and enter a value in the **Tolerance** field. For

more information regarding these options, refer to the section on **Shapefile Synchronization Options**. Click the **Next** button to proceed to the **Shapefile Link Editors**.

6. You will be presented with an **Import Shapefile Link Editor** for each element type you chose to import. Perform the following steps for each **Import Shapefile Link Editor**:
   - Enter the name of the Shapefile to which you wish to connect for the specified element type. Click the **ellipsis (…)** button to interactively browse for and select your Shapefile.
   - Choose the **Key\Label** field to define the column in the Shapefile that maps to the element labels in the model.
   - Define as many field links as you want by selecting the model attribute and the associated **Shapefile** column and unit if appropriate. Use the **Select** button for making the selection process more efficient. Click the **Next** button.
7. Check the **Synchronize Shapefile Connection** box if you wish to synchronize the connection immediately upon clicking the **Finished** button.
8. If the **Synchronize Shapefile Connection** box is checked, choose whether you want to Synchronize In to the model from a Shapefile, or Synchronize Out to the Shapefile from the model.
9. Click the **Finished** button to synchronize the connection if the **Synchronize Shapefile Connection** box is checked, or to return to the **Shapefile Connection Manager**.

# Chapter 14

# Exchanging Data with CAD Software and Autodesk Civil Design

## 14.1 AutoCAD Polyline to Pipe Conversion

This feature allows you to quickly construct a network based on the entities contained in an AutoCAD drawing. Although this feature is called Polyline to Pipe, Line and Block entities can be converted as well. Polylines and Lines can be converted to pipes and Blocks can be converted to any available node type.

Building a model based on graphical elements can be an error-prone process. This is due to the fact that a drawing can appear to be correct visually, but may contain problems that are not readily apparent. For example, what appears to be a single line in a drawing could in fact be made up of many line segments, or it could be made up of two lines, one directly on top of another.

To help alleviate some of the problems that you may encounter during the import process, a comprehensive drawing review is also performed. During the conversion process, the network is analyzed and potential problems are flagged for review. After performing the conversion, the **Drawing Review** window will allow you to navigate to and fix any problems that are encountered.

**The Polyline to Pipe conversion cannot be undone. Be sure to save your project before you begin.**

**You can import entities into an existing project. Polylines will automatically be connected to nodes within the specified Tolerance. You can add nodes to your project prior to performing the import.**

**Stand-Alone mode - You should take some time to clean up your AutoCAD drawing prior to performing the conversion. Look for entities that should not be converted, such as leader lines, and move them to their own layer. Turn off layers that you do not wish to convert. Do a quick review of your drawing and correct any potential conversion problems that you may find.**

**After performing the conversion, we recommend that you use the converted file as a DXF Background. This will greatly enhance your review process. If you change the entities in your background drawing to a gray color from within AutoCAD, it will make it easier to distinguish between foreground elements and background entities.**

**AutoCAD mode - You can interactively convert individual entities to pipes by using the Layout Tool.**

**SewerCAD only - When importing a drawing that contains both pressure and gravity pipes you should import your system in two passes. First, import the layer(s) that contain Gravity pipes, then import the layer(s) that contain Pressure pipes. On the first pass create Manholes at pipe endpoints, and on the second pass create Pressure Junctions at pipe endpoints. This will ensure that the correct element will be added where your system transitions from gravity to pressure. Refer to the related information section on "Converting your drawing in multiple passes" for more information.**

### 14.1.1 Polyline to Pipe Wizard

The **Polyline to Pipe Wizard** will guide you step-by-step through the process of converting your entities to elements.

- **Step 1** - The import behavior depends on the mode in which you are working:
  - *Stand-Alone* - Specify the .DXF file that you would like to import.
  - *AutoCAD* - This step is skipped. You will be asked to select the entities to convert before accessing the Wizard.
- **Step 2** - Specify the polyline to pipe conversion options.
- **Step 3** - Specify how T-intersections are to be handled.
- **Step 4** - Specify how blocks should be converted (for .DXF files that contain blocks).
- **Step 5** - Configure prototypes.
- **Step 6** - Specify the layers to be imported.

To access the **Polyline to Pipe Wizard**:

| | |
|---|---|
| *Stand-Alone* | Select **File / Import / Polyline to Pipe** from the main menu. |
| *AutoCAD* | Select **Edit / Change Entities to Pipes** from the main menu. |

### 14.1.2 Polyline to Pipe Wizard - Step 1 (Stand-Alone mode only)

This step allows you to specify the .DXF file to be imported.

**If you are running in AutoCAD mode, this step will be skipped. AutoCAD mode users will be asked to select the entities to be converted before accessing the Polyline to Pipe wizard.**

- **DXF Filename** - Specify the name of the .DXF file you would like to import. Use the **Browse** button to select the file interactively.
- **DXF Unit** - Specify the .DXF conversion unit (the unit that your .DXF file is in). For example, if your drawing is in SI units, specify meters (m). If your drawing is in architectural units, specify inches (in).

### 14.1.3 Polyline to Pipe Wizard - Step 2

This step allows you to specify the following Polyline to Pipe conversion options:

- **Connectivity tolerance** - Polylines whose endpoints fall within the specified tolerance will be connected to the same node. A default tolerance is supplied based on the current scale. This is generally a good starting point, but you may wish to increase or decrease this default tolerance depending on your particular drawing. If you complete the conversion process and find that the tolerance was not correct (pipes that should be connected were not, or vise versa), you may wish to repeat the conversion process using a new tolerance.

- **Specifying which entities to convert -** You can optionally convert Polylines, Lines, or both. You generally want to convert both Polylines and Lines. However, if your drawing is set up so that Polylines are always used to represent pipes and Lines are used for annotation purposes, you may wish to convert only Polylines.
- **Handling missing nodes at polyline endpoints** - A pipe can only be created if there is a node at both endpoints. If a node cannot be found at a polyline endpoint, a node must be added. Otherwise, the pipe cannot be converted. This option allows you to specify whether a node is created, and, if so, the default type of element to create.

In general, you will want to create a default node at polyline endpoints. However, if your network already contains nodes at polyline endpoints, or if your drawing contains blocks at polyline endpoints that are to be converted to nodes, you may wish to specify that the polyline not be converted. Polylines that cannot be converted, because one or both end nodes are missing, will be flagged for review at the end of the conversion process.

**If the conversion does not yield the desired results, you can repeat the conversion process using different settings. Be sure to save your project before performing the conversion.**

### 14.1.4 Polyline to Pipe Wizard - Step 3

This step allows you to specify how T-intersections (pipe split candidates) should be handled.

Nodes that fall within the specified tolerance of a pipe are referred to as *pipe-split candidates.* There are two ways to handle these:

- **Join the pipes at the intersection -** The pipe-split candidate will be used to split the intersecting pipe.
- **Do not join the intersecting pipes -** Pipe-split candidates will be flagged for later review using the **Drawing Review** window.

**The tolerance that you specify in Step 2 will also be used for T-intersection processing.**

### 14.1.5 Polyline to Pipe Wizard - Step 4 (for .DXF files that contain blocks)

If your AutoCAD drawing contains blocks, this step will appear, allowing you to convert AutoCAD blocks, if desired.

If you would like to convert blocks to nodes, activate the **Yes** toggle. A table with two columns will appear, allowing you to map the AutoCAD blocks you would like to convert to any of the available node element types. The **AutoCAD block** column provides you with a list of available blocks to convert. The **Element** column provides you with a list of available node element types.

For each AutoCAD block you would like to convert, specify the type of node element you would like to create.

**When you select an AutoCAD block, the preview pane will display the graphical representation of that block. This step will be skipped if there are no AutoCAD Blocks in your drawing.**

### 14.1.6 Polyline to Pipe Wizard - Step 5

Before performing the conversion, you may wish to configure your prototypes with default data. During the conversion process, elements will be created using the specified defaults.

Click a button to configure the defaults for the associated element.

### 14.1.7 Polyline to Pipe Wizard - Step 6

Specify the layers that contain the entities you would like to convert. Use the **Preview Drawing** button to preview the elements on the selected layers. This step can be used in conjunction with the Prototype step to allow you to convert your drawing in multiple passes.

**It is recommended that you process your drawing prior to performing the import. If your drawing contains layers that you do not wish to import, turn them off from within AutoCAD and elements on those layers will be ignored during the import process.**

### 14.1.8 Drawing Preview

Use the **Preview Drawing** button to view the elements in the .DFX file that will be converted.

Next to the **Preview Drawing** button is a checkbox labeled **Only include elements that will be converted**.

Turn the toggle on to preview the entities that will be converted. The entities to be converted are based on the settings you specified in the **Polyline to Pipe Wizard**, such as type of line entities, blocks, and layers to be converted.

Turn the toggle off to preview all entities.

### 14.1.9 Polyline Conversion Problem Dialog

This feature is present in Stand-Alone mode only. This dialog displays the reason that a polyline was not converted after running the **Polyline to Pipe Wizard**.

### 14.1.10 Converting your Drawing in Multiple Passes

Depending on how your drawing layers are set up, you may be able to save yourself a considerable amount of data entry time by converting your drawing in multiple passes.

For example, if your 12-inch pipes are located on a "12InchPipes" layer, 18-inch pipes are on a "18InchPipes" layer, etc., you can import layers one at a time. Just set up your prototypes prior to importing that layer.

To assist you in this process, your conversion settings will be retained between imports. Therefore, on subsequent passes you will simply need to revise your prototypes and specify the next layer to be imported.

This same technique can be used when importing blocks.

## 14.2 Land Development Desktop - Civil Design Connection

### 14.2.1 Land Development Desktop Import Wizard

This import procedure allows you to construct a model based on your Land Development Desktop project data. The **Import Wizard** will guide you step-by-step through the process of importing data from the *Civil Design* module of AutoCAD’s Land Development Desktop.

Following are the basic steps for importing a Land Development Desktop file containing the pipe data:

- **File Import Settings** - Select the pipeworks.mdb file to import, the unit system in which it is stored, and the connectivity tolerance.
- **Runs to Import** - Pick the runs that you would like to import from the list of available runs in your Land Development Desktop project file.
- **Import Structure Mappings** - Specify the mapping between Land Development Desktop structure names and this application's node types.

> **You can import a Land Development Desktop data file into an existing project to add new pipes and structures.**

### 14.2.2 File Import Settings

**File Name** - Enter the path and name of the file to import, or use the **Browse** button to select it interactively. The *Civil Design* data is stored in a file named *pipeworks.mdb* located in the *pipewks* directory of your Land Development Desktop project directory. For example, if your project is named "*myproject*", your data may be stored in *C:\Land Projects\myproject\pipewks\pipeworks.mdb*.

**Connectivity Tolerance** - Runs whose end nodes fall within the specified tolerance will be connected to the same node. A default tolerance is supplied; you may wish to increase or decrease this tolerance depending on your particular project. If you complete the import process and find that the tolerance was not appropriate (pipes that should be connected are not, or vice versa), you may wish to repeat the conversion process using a new tolerance.

**Land Development Desktop Unit System** - Specify whether the Land Development Desktop project you wish to import is in the SI or US Customary unit system. The imported data will automatically be converted to your current project unit system.

### 14.2.3 Runs to Import

Land Development Desktop maintains network connectivity based on runs. Runs consist of reaches, and reaches consist of an upstream node and a downstream pipe. A list of all runs present in your Land Development Desktop file will be displayed. Toggle the check boxes to specify the runs you would like to import.

Remember that you can always import more runs into your project later.

### 14.2.4 Import Structure Mappings

A list of the structure labels in your Land Development Desktop file will be displayed. For each structure label, specify the type of node element that you would like to create.

### 14.2.5 Land Development Desktop Export Wizard

The **Land Development Desktop - Civil Design Export Wizard** will guide you step-by-step through the process of exporting part (or all) of your network to a database file. That database file can then be imported into your Land Development Desktop project (using the *Civil Design* module).

Following are the basic steps for exporting data to a Land Development Desktop file:

- **File Export Settings** - Select the name of the database file to which you would like to export your data, and the unit system in which your Land Development Desktop data is stored.
- **Runs to Export** - Specify runs containing the elements you would like to export.

- **Export Structure Mappings** - Specify the mapping between this application's node types and your Land Development Desktop Structure names.

After exporting your data, you can use the Land Development Desktop to import that data into your *Civil Design* project. From within the Land Development Desktop, use the **Pipes / Import-Export Run / Import DB** menu item to import the desired runs into your Land Development Desktop project. Use the **Conceptual Plan / Import Run** command from the **Pipes** main menu to display each run in a plan view. Then use the Land Development Desktop functions to generate Plan and Profile sheets as well as construction drawings.

**Only Gravity Elements can be exported.**

By default, the Land Development Desktop Export Wizard creates an Office 97 Access file. If you want to create an Office 2000 Access file, then you need to edit (with a text editor) the HAESTAD.INI file located in your HAESTAD directory and replace the line:

PipeworksDatabaseFormat=3

With:

PipeworksDatabaseFormat=0

Basically a value of 3 results in the program creating an Office 97 Access file (Jet versions), whereas a value of 0 will have the program generate the default Office 2000 Access file.

### 14.2.6 File Export Settings

**File Name** - Enter the name of the database file to which you would like to export your data, or use the **Browse** button to interactively specify the file name.

**Land Development Desktop Unit System** - Specify whether you want to export data to the Land Development Desktop in the SI unit system, or in the US customary unit system.

### 14.2.7 Runs to Export

Land Development Desktop maintains network connectivity based on runs. Runs consist of reaches, and reaches consist of an upstream and a downstream node. This export command allows you to specify the runs to be exported. You can either specify the runs interactively, or you can automatically generate a list of runs to represent the entire network, using the following commands:

- Add Runs
- Edit Runs
- Delete Runs
- Initialize Run List

**Only gravity elements can be exported.**

### 14.2.8 Add/Edit Pipe Run

Specify the description (or label) for the pipe run, and specify the upstream and downstream nodes that define that run.

### 14.2.9 Delete Runs

Removes the selected run from the list of runs to export.

### 14.2.10 Initialize Run List

Use this option to automatically generate a list of runs to represent your entire network. When you click the **Initialize** button, the **Element Labeling** dialog will appear, allowing you to customize the automatic run label generation.

After building the run list, modify the default runs using the **Add**, **Edit** and **Delete** buttons.

### 14.2.11 Automatic Element Labeling

The **Element Labeling** dialog is used to specify the format of the numbering automatically generated by the runs using the **Initialize** button.

**Next** - Integer you want to use as the starting value for the ID number portion of the run label.

**Increment** - Integer that you want to be added to the current ID number to build the next run label.

**Prefix** - Letters or numbers that you want to appear in front of the ID number for the run labels.

**Digits** - Total number of digits you want the ID number to have.

**Suffix** - Letters or numbers that you want to appear after the ID number for the run labels.

**Preview** - An example of what the label will look like, based on the information you have entered in the fields described above.

### 14.2.12 Export Structure Mapping

A list of all node types to be exported will be displayed. For each node type, select the **Structure** label you want to export to the Land Development Desktop database file. The labels should correspond to those you are using in your **Land Development Desktop Structure Library Editor**.

## 14.3 Import/Export of DXF Files

### 14.3.1 Import a DXF from AutoCAD or MicroStation

To import background graphics in Stand-Alone mode from another drafting program, you must first export a .DXF file from your CAD program. This step is usually as simple as selecting an item from a pull-down menu in that program, such as **File\Export\As DXF**, or similar command. Once the .DXF file has been created, it can be imported into this program as follows:

1. Select the File\Import\DXF Background command from the pull-down menu to access the Import DXF File dialog.
2. Select the .DXF file you wish to import, and click the Open button.

### 14.3.2 Exporting a DXF file

To export the drawing plan view, select **File / Export / DXF file** from the pull-down menu.

**You will be able to redefine all elements, except pipes, as blocks in AutoCAD. Pipes will be exported as polylines, so you will be able to set their line weight in AutoCAD.**

### 14.3.3 Redefining SewerCAD Blocks in AutoCAD

When exporting a .DXF file from SewerCAD, pipes will be exported as POLYLINE entities allowing you to change the line weights in AutoCAD. Nodes will be exported as BLOCK entities named after the element, such as MANHOLE, PUMP, WETWELL, and OUTLET. This allows you to redefine them in AutoCAD.

If you would like to change the appearance of these blocks in your AutoCAD drawing, you can redefine them by performing the following steps:

1. Start AutoCAD and create separate drawing files named MANHOLE.DWG, PUMP.DWG, WETWELL.DWG, OUTLET.DWG, etc. Save these drawings in your AutoCAD directory.
2. Open the existing drawing that contains the SewerCAD blocks.
3. At the AutoCAD command prompt, type **MINSERT** and press **Enter**.
4. At the "Block Name:" prompt, type **MANHOLE=C:MANHOLE.DWG** and press **Enter**.
5. You may be prompted to verify that you want to redefine the block. Answer "Yes."
6. At this point, the block has been redefined and you can cancel this command.
7. Repeat these steps for PUMP, WETWELL, OUTLET, etc.

Refer to your AutoCAD documentation for more information on Redefining Blocks.

### 14.3.4 Advanced DXF Import Techniques

If you would like to import a SewerCAD .DXF file into an existing AutoCAD drawing file, you will have to perform a couple of preliminary steps.

1. In your existing drawing at the AutoCAD command prompt, type **(regapp "SWRC")** and press **Enter**. This will register the SewerCAD application id, so be sure to include the parenthesis.
2. Define blocks named MANHOLE, PUMP, WETWELL, OUTLET, etc.

You are now ready to import a SewerCAD .DXF file into your existing AutoCAD drawing.

To save time, you can perform the above steps in a new AutoCAD drawing file and save it with the name SewerCAD.DWG. Now, instead of performing the above steps, simply insert this new drawing into your existing drawing file immediately before importing a SewerCAD .DXF file.

Refer to your AutoCAD documentation for more information on importing .DXF files.

# Chapter 15

# Additional Features of the AutoCAD Version

## 15.1 Overview

SewerCAD features optional support for AutoCAD integration. You can determine if you have purchased AutoCAD functionality for SewerCAD by using the **Help / About SewerCAD** menu option. Click the **Registration** button to view the feature options that have been purchased with your application license. If AutoCAD support is enabled, you will be able to run your SewerCAD application in both AutoCAD and Stand-Alone mode.

The AutoCAD functionality has been implemented to be essentially identical to that offered with the Stand-Alone product. Once you obtain familiarity with the Stand-Alone mode, you will encounter few difficulties utilizing the product in AutoCAD mode.

In AutoCAD mode, you will have access to the full range of functionality available in the AutoCAD design and drafting environment. The standard environment is extended and enhanced by an AutoCAD ObjectARX SewerCAD client layer that allows you to create, view, and edit the native SewerCAD network model while in AutoCAD.

Some of the advantages of working in AutoCAD mode include:

- Layout sanitary sewer pipelines and structures in fully scaled mode in the same design and drafting environment that you use to develop your engineering plans. You will have access to any other third party applications that you currently use, along with any custom LISP, ARX, or VBA applications that you have developed.
- Use native AutoCAD insertion snaps to precisely position SewerCAD elements with respect to other entities in the AutoCAD drawing.
- Use native AutoCAD commands such as ERASE, MOVE, and ROTATE on SewerCAD model entities with automatic update and synchronization with the model database.
- Output profiles and schematics to your AutoCAD drawing.
- Control destination layers for model elements and associated label text and annotation, giving you control over styles, linetypes, and visibility of model elements.

## 15.2 SewerCAD Custom AutoCAD Entities (AutoCAD Mode)

This feature is only available in the AutoCAD mode of SewerCAD. The primary AutoCAD-based SewerCAD element entities - gravity pipes, pressure pipes, manholes, junction chambers, wet wells, pressure junctions and outlets - are all implemented using ObjectARX custom objects. Thus, they are vested with a specialized "model awareness," which ensures that any editing actions that you make will result in an appropriate update of the model database.

This means that you can perform standard AutoCAD commands as you normally would, and the model database will be updated automatically to reflect these changes.

This also means that the model will enforce the integrity of the network topological state. So, if you delete a nodal element such as a manhole, its connecting pipes will also be deleted.

Using ObjectARX enables the implementation of highly specialized editing actions that are not available with standard AutoCAD entities. Two examples of this specialized behavior are element morphs, which change a node from one element type to another, and pipe splits. Again, these modifications will trigger an automatic update of the model network topology and associated element properties.

Using ObjectARX technology ensures that the database will be adjusted and maintained during Undo and Redo transactions.

A custom model element has certain native text entities associated with it for displaying label and annotated property values. These associated label and annotation entities may be edited separately from the model element itself. However, most drawing edits made directly to a model element will also be applied to its associated label and annotation entities. Thus, if you drag an element to a new location, the annotation and label locations will update as well.

## 15.3 AutoCAD Environment

### 15.3.1 AutoCAD Mode Graphical Layout

In AutoCAD mode, Haestad Methods' products provide a set of extended options and functionality beyond those available in Stand-Alone mode. This additional functionality provides enhanced user control over general application settings and options and extends the command set, giving the user control over the display of model elements within AutoCAD.

Key differences between AutoCAD and Stand-Alone mode include:

- Element editing functionality has been extended by adding the **Scale Elements** and **Rotate Labels** commands, accessible under the **Edit / Modify Elements** pull-down menu, and the **Change Width** command under the **Edit / Pressure Pipes** pull-down menu.
- You can control the appearance and destination of all model elements using the **Element Properties** command under the **Tools** pull-down menu. For example, you can assign a specific layer for all outlets, as well as assign the label and annotation text style to be applied.
- Though right-click context menus are now standard with AutoCAD 2000 and 2000i, a **Right-Click Context Menu Option** has been added to provide optional conformity with the Stand-Alone mode of operation in AutoCAD R14.

### 15.3.2 Toolbars

In AutoCAD mode, the following toolbars are available:

- **Command Tools -** Enables the **Command Toolbar** for quick access to the main commands, including computations, tables, graphic reports, Quick View, and direct access to the Haestad Methods Web Site.
- **Layout Tools** - Enables the Layout Toolbar for access to the Tool Palette.
- **Analysis Toolbar -** Enables the **Analysis Toolbar** to control the current scenario and provide quick access to the **Scenario Manager** and **Cost Manager**, as well as time and animation controls.

To toggle the display of the Haestad Methods command and layout toolbars, select **View / Toolbars** from the pull-down menu.

### 15.3.3 Drawing Setup

When working in the AutoCAD mode, you may work with Haestad Methods' products in many different AutoCAD scales and settings. However, Haestad Methods' product elements can only be created and edited in model space.

### 15.3.4 Symbol Visibility

This is only available in the AutoCAD mode of SewerCAD. You can control display of element labels using the **Show Labels** checkbox, found by selecting **Tools / Options** from the pull-down menu and choosing the **Drawing** tab.

The following commands allow you to customize the drawing by turning the visibility of flow arrows and/or labels on or off without accessing the **Options** dialog.

- To turn on the element labels, type: **SWRCLABELSON**
- To turn them off, type: **SWRCLABELSOFF**

**In AutoCAD, it is possible to delete element label text using the ERASE command. You should not use ERASE to control visibility of labels. Instead, the command above. If you desire to control the visibility of a selected group of element labels, you should move them to another layer that can be frozen or turned off.**

**See Rebuild Figure Labels for more information on restoring labels that have been erased using the native AutoCAD command.**

### 15.3.5 Rebuild Figure Labels

This is only available in the AutoCAD mode of SewerCAD. It is possible to delete element label text entities. Element labels which have been erased can be selectively undeleted using the command **SWRCREBUILDLABELS**.

## 15.4 AutoCAD Project Files

When using SewerCAD in AutoCAD mode, it is important to remember that two files (file extensions shown in parentheses) fundamentally define a SewerCAD model project:

- Drawing File (.DWG) - The AutoCAD drawing file contains the custom entities that define the model, in addition to the planimetric base drawing information that serves as the model background.
- Model File (.SWR) - The native SewerCAD model database file contains all the element properties along with other important model data. SewerCAD .SWR files can be loaded and run using the Stand-Alone mode. These files may be copied and sent to other SewerCAD users who are interested in extending or running your project. This is the most important file for the SewerCAD model.

**The two files will have the same base name. It is important to understand that simply archiving the drawing file is not sufficient to reproduce the model. You must also preserve the associated .SWR file.**

Since the .SWR file can be run and modified separately from the .DWG file using Stand-Alone mode, it is quite possible for the two files to get out of sync. Should you ever modify the model in Stand-Alone mode and then later load the AutoCAD .DWG file, the SewerCAD program will compare file dates and automatically invoke its built-in AutoCAD Synchronization routine.

### 15.4.1 Drawing Synchronization

Whenever you open a SewerCAD-based drawing file in AutoCAD, the SewerCAD model server will start. The first thing that the application will do is load the associated SewerCAD database (.SWR) file. If the time stamps of the drawing and database files are different, SewerCAD will automatically invoke its synchronization check. This protects against corruption that might otherwise occur from separately editing the SewerCAD database file in Stand-Alone mode or editing proxy elements at an AutoCAD station where the SewerCAD application is not loaded.

The synchronization check will occur in two stages:

1. During the first stage of the check, SewerCAD will review all the drawing model elements and compare their state with that held in the server model. Any differences it discovers during this check will be listed. SewerCAD enforces network topological consistency between the server and the drawing state. If model elements have been deleted or added in the .SWR file during a Stand-Alone session, or if proxy elements have been deleted, the application will force the drawing to be consistent with the native database by restoring or removing any missing or excess drawing custom entities.
2. After network topology has been synchronized, the application will compare other model and drawing state such as location, labels, and flow directions. Again, it will list any differences between the drawing client and server data, but a message box will pop up giving you an opportunity to indicate which state, drawing or model server, should be adopted during the second stage of synchronization.

You can run the synchronization check at any time using the command **SWRCSYNCSERVER**.

### 15.4.2 Saving the Drawing as Drawing*.dwg

AutoCAD uses Drawing*.dwg as its default drawing name. Saving your drawing as the default AutoCAD drawing name (for instance Drawing1.dwg) should be avoided, as it makes overwriting model data very likely. When you first start AutoCAD, the new empty drawing is titled Drawing*.dwg, regardless of whether one exists in the default directory. Since Haestad Methods' modeling products create model databases associated with the AutoCAD drawing, the use of Drawing*.dwg as the saved name puts you at risk of getting the AutoCAD drawing and Haestad Methods modeling files out of sync.

**If this situation is forced to occur (save on quit for example), simply restart AutoCAD, use the Open command to open the Drawing*.dwg file from its saved location, and use the Save As command to save the drawing and model data to a different name.**

## 15.5 Element Properties

When working in the AutoCAD mode, this feature will display a tabbed dialog with tables containing different model element types and their associated properties, along with the properties of the element's layer, label, and annotation. To modify an attribute, double-click each associated grid cell. Setting changes made in this dialog will be used for any newly created elements. Property changes will be performed on all elements of the given type. If the **Apply to Existing Object** box is checked, modifications made in this dialog are performed on a global basis. To restrict global changes to a certain layer for a particular element type, use the "*current*" option setting for the attribute of interest.

To change the layer, label, or annotation of an element, select **Tools / Element Properties** from the pull-down menu.

### 15.5.1 Select Layer

When running in AutoCAD mode, this dialog appears when you double-click the layer name ("*current*" by default) in the **Layer** column of the **Element Properties** dialog. This is accessed by selecting **Tools / Element Properties** from the pull-down menu. It displays a list of the available layers and their properties from the current AutoCAD drawing. Click the appropriate field to select a layer. The "*current*" option will use whatever layer is set to current in your AutoCAD drawing.

### 15.5.2 Select Text Style

When running in AutoCAD mode, this dialog appears when you double-click the text style name ("*current*" by default) in the **Text Style** column of the **Labels** and **Annotation** tabs of the **Element Properties** dialog. This is accessed by selecting **Tools / Element Properties** from the pull-down menu. It displays a list of the available text styles and their properties from the current AutoCAD drawing. Click the appropriate field to select a text style. The "*current*" option will use whatever text style is set to current in your AutoCAD drawing.

## 15.6 Working with Elements

### 15.6.1 Edit Element

In AutoCAD mode, this menu selection will open an element editor for any specific element. Select **Edit / Edit Element**, then select an element. This command is also available by choosing the **Select** tool, then clicking an element in the drawing pane.

The **Edit Element** command works with the current selection to allow you to generate filtered reports. Refer to Selecting Elements (AutoCAD Mode) for more information on working with selections.

### 15.6.2 Edit Elements

In AutoCAD mode, this menu command is used to open a spreadsheet FlexTable editor or a selection of one or more network figures. You are prompted to select figures on which to build a table.

### 15.6.3 Deleting Elements

In AutoCAD mode, this command removes all elements in the current selection. Refer to Selecting Elements (AutoCAD Mode) for more information on working with selections.

### 15.6.4 Modifying Elements

#### Modify Elements

In AutoCAD mode, these commands are selected from the **Edit** pull-down menu. They are used for scaling and rotating model entities.

#### Scale Elements

In AutoCAD mode, this menu selection resizes an element based upon a scale factor. After choosing this command, select an element or group of elements, and enter the scale factor to be applied.

To access the **Scale Elements** command, select **Modify Elements** from the **Edit** pull-down menu.

#### Rotate Labels

In AutoCAD mode, this menu selection rotates the figure label. After choosing this command, select an element or group of elements, and enter the desired rotation in degrees.

To access the **Rotate Labels** command, select **Modify Elements** from the **Edit** pull-down menu.

#### Modify Pressure Pipes

Pressure pipes have layout characteristics that are distinct from gravity pipes. The main difference is that SewerCAD pressure pipes may follow a non-linear alignment (since in pressure systems, unlike gravity runs, minor losses can be safely lumped with friction losses without significantly affecting model accuracy). SewerCAD uses specialized commands for editing pressure pipes in AutoCAD.

- **Insert Bend** - Use this command to add a bend to a pressure pipe. In AutoCAD, you will be prompted to select a pipe to bend. Select the pipe and then select the location where you want the bend to appear. The pipe alignment will automatically conform to this location.
- **Remove Bend** - Use this command to remove a specific bend from a pressure pipe. In AutoCAD, you will be prompted to select a pipe and, subsequently, the specific bend to remove.
- **Remove All Bends** - Use this command to completely straighten a pressure pipe that contains bends. In AutoCAD, you will be prompted to select a pipe and all bends will disappear.
- **Change Widths** - This menu selection changes pipe widths. After choosing this command, select a pipe or group of pipes and enter the desired width. Note that the width entered is equivalent to the AutoCAD polyline width.

## 15.7 Working with Elements Using AutoCAD Commands

### 15.7.1 AutoCAD Commands

When running in AutoCAD mode, Haestad Methods' products make use of all the advantages that AutoCAD has, such as plotting capabilities and snap features. Additionally, AutoCAD commands can be used as you would with any design project. For example, Haestad Methods' elements and annotation can be manipulated using common AutoCAD commands.

### 15.7.2 Explode Elements

In AutoCAD mode, running the AutoCAD **Explode** command will transform all Haestad Methods custom entities into equivalent AutoCAD native entities. When a Haestad Methods custom entity is exploded, all associated database information is lost. Be certain to save the exploded drawing under a separate filename.

Use **Explode** to render a drawing for finalizing exhibits and publishing maps of the model network. You can also deliver exploded drawings to clients or other individuals who do not own a Haestad Methods Product license, since a fully exploded drawing will not be comprised of any ObjectARX proxy objects. See Working with Proxies for more information on this topic.

### 15.7.3 Moving Elements

When using AutoCAD mode, the AutoCAD commands **Move**, **Scale**, **Rotate**, **Mirror**, and **Array** can be used to move elements. Refer to Selecting Elements for more details on this topic.

To move a node, execute the AutoCAD command either by typing it at the command prompt or selecting it from the pull-down menu. Follow the AutoCAD prompts, and the node and its associated label will move together. The connecting pipes will shrink or stretch depending on the new location of the node.

### 15.7.4 Moving Element Labels

When using AutoCAD mode, the AutoCAD commands **Move**, **Scale**, **Rotate**, **Mirror**, and **Array** can be used to move element text labels. Refer to the help topics Selecting Elements and Working with Selections in AutoCAD.

To move an element text label separately from the element, click the element label you wish to move. The grips will appear for the label. Execute the AutoCAD command either by typing it at the command prompt or selecting it off the tool palette. Follow the AutoCAD prompt, and the label will be moved without the element.

### 15.7.5 Snap Menu

When using AutoCAD mode, the **Snap** menu is a standard AutoCAD menu that provides options for picking an exact location of an object. Refer to the standard AutoCAD help documentation for more information.

## 15.8 Undo / Redo

### 15.8.1 Undo and Redo Operations in AutoCAD

In AutoCAD, you have two types of Undo/Redo available to you. From the **Edit** menu, you have access to SewerCAD Undo and Redo. Alternatively, you can perform the native AutoCAD Undo and Redo by typing at the AutoCAD command line. The implementations of the two different operation types are quite distinct.

The menu-based undo and redo commands operate exclusively on SewerCAD elements by invoking the commands directly on the model server. The main advantage of using the specialized command is that you will have unlimited undo and redo levels. This is an important difference, since in layout or editing it is quite useful to be able to safely undo back and then redo ahead an arbitrary number of transactions. If you use the native AutoCAD undo, you are limited to a single redo level. The SewerCAD undo/redo is also faster than the native AutoCAD undo/redo. If you are rolling back SewerCAD model edits, it is recommended that you use the menu-based undo/redo implementation.

Whenever you invoke a native AutoCAD undo, the model server will be notified when any SewerCAD entities are affected by the operation. SewerCAD will then synchronize the model to the drawing state. Wherever possible, the model will seek to map the undo/redo invocation onto the model server's managed command history. If the drawing state is not consistent with any pending undo or redo transactions held by the server, SewerCAD will flush the command history. In this case, the model will synchronize the drawing and server models in a rigorous fashion.

It is important to note that if you undo using the AutoCAD command and you end up restoring SewerCAD elements that have been previously deleted, morphed, or split, some model state such as diameter or elevations may be lost, even though the locational and topological state is fully consistent. This will only happen in situations where the SewerCAD command history has been flushed. In such cases, you will be warned to check your data carefully.

## 15.9 Converting Native AutoCAD Entities to SewerCAD Elements

### 15.9.1 Converting Native AutoCAD Entities

SewerCAD features powerful tools dedicated to assisting the user in building SewerCAD models from existing AutoCAD drawing information. In addition to the standard ESRI Shapefile conversion options, there are two specific commands available in the AutoCAD platform that will be especially useful to the AutoCAD modeler:

- Layout Pipe Using Entity
- Change Entities to Pipes

### 15.9.2 Layout Pipe Using Entity

In addition to the standard options available under the Pipe layout command (accessed by clicking on the button in the SewerCAD Tools toolbar or by selecting using **Tools / Layout / Pipe** from the pull-down menu), you may elect to use an existing AutoCAD line, polyline, or arc as a template to define an equivalent SewerCAD pipe or series of pipes.

While you are in the **Pipe Layout** command, you may invoke the Entity conversion option by using the **'Entity'** keyword or by selecting **'Entity'** from the right-mouse button context menu. Once selected, you will be prompted to select an entity to use as a basis for a new pipe and conditionally specify the type of nodal SewerCAD element(s) to use at each end of the pipe.

**This command is extremely useful for constructing pressure pipes that follow a curved alignment. In these cases use an arc as the defining template entity for the pipe creation.**

### 15.9.3 Change Entities to Pipes

This special AutoCAD command allows you to use a selection of AutoCAD entities - arcs, lines, polylines, and blocks - as a defining template set for the creation of equivalent SewerCAD elements. This command performs the element generation in batch fashion. You are prompted for the selection of entities to convert, and the selection is followed by the **Polyline To Pipe Conversion Wizard** that leads you through a sequence of steps defining the basis of the batch conversion. The actual steps to be followed in the Wizard are fully described in the AutoCAD Polyline to Pipe Conversion topic in the "Exchanging Data with CAD software and AutoDesk Civil Design" chapter.

**It is important to note that this is an automated batch process which requires some care and attention with respect to the selection set that is going to be used as a basis for generating actual SewerCAD model elements. Specifically, you probably want to process gravity sub-network elements separately from pressure sub-network elements. It is also desirable to select like-sized pipe elements during each pass. This way, you can use the prototyping capabilities to their greatest advantage. A little time spent in planning and strategizing a series of individual conversion steps will go a long way toward preventing confusion, which could necessitate later re-conversions.**

## 15.10 Special Considerations

### 15.10.1 Import SewerCAD

This is only available in the AutoCAD mode. This command imports a selected SewerCAD data (.SWR) file for use in the current drawing. The new project file now corresponds to the drawing name, i.e. CurrentDrawingName.SWR. Whenever you save changes to the network model through SewerCAD, the new associated .SWR data file is updated and can be loaded into SewerCAD 5.0 or higher.

To import a SewerCAD model into AutoCAD, select **File / Import / SewerCAD**.

### 15.10.2 Working with Proxies

If you open a SewerCAD drawing file on an AutoCAD workstation that does not have the SewerCAD application installed, you will get an AutoCAD **Proxy Information** message box. This is because the executable logic for managing the AutoCAD entities is not available, and the SewerCAD modeling elements are not associated with the SewerCAD native database.

SewerCAD proxy objects can be moved and erased. But doing so will put the drawing state out of sync with the model database should the drawing be saved with its original name. If this happens, and you later reload the drawing on an AutoCAD station that is running a SewerCAD application, the application will automatically attempt to reconcile any differences it finds by automatically loading its Database Synchronization routine.

Notes

# Appendix A

# Frequently Asked Questions

## A .1 Overview

To make your work easier, SewerCAD and the Help system are designed to be used together.

If you have a high resolution display monitor, you will probably find it helpful to size the frames of both the program and the **Help** windows so that they fit side by side. Then, while using the program, you can use the **Help** button in any dialog to update the **Help** window.

If the information you need is not on the **How Do I** page, click the **Search** button at the top of the **Help** window to access the search index.

## A .2 How Do I Control Element and Label Sizing?

To change the size of element symbols and labels:

1. Select **Tools / Options** from the pull-down menus, and select the Drawing tab.
2. In the **Annotation Multipliers** group, change the **Symbol Size Multiplier** to modify the element size, and the **Text Height Multiplier** to modify the label size. Smaller numbers will make the element symbols and text decrease in size.

These changes will affect all symbols and text, including color coding legends, but will not have any effect on pipe lengths.

## A .3 How Do I Reuse Deleted Element Labels?

To make the program reuse the label for a deleted element:

1. Select **Tools / Element Labeling** from the pull-down menu.
2. Enter the ID number for the deleted element in the **Next** field for the appropriate type of element.
3. Click **OK**.
4. Add a new element to the drawing.

## A .4 How Do I Color Code Elements?

To color code elements:

1. Select **Tools / Color Coding...** from the pull-down menu, or click the **Color Coding** (rainbow) button on the toolbar.
2. In the **Color Coding** dialog, select the attribute you would like to color code.

3. Click the **Initialize** button to automatically build a range of colors. You may decide to modify these default ranges.
4. Click **OK** to color code the drawing.

All link or node elements and their labels will be colored based on the specified ranges. You can also use the **Initialize** button to quickly set up and modify **Color Coding Options**. A **Color Coding Legend** may be inserted into the drawing by using the **Legend** tool located on the **Tool Palette**.

## A .5 How Do I Remove Color Coding from Labels Imported from Pre-v3.5 AutoCAD Files?

Due to popular request, Haestad Methods has implemented the separation of elements and their labels. This gives you much more control over the placement and formatting of the labels, in addition to resolving the problem of color coding labels with elements. However, if you open an old drawing (version 3.1 or earlier) with existing color coding on the labels, this color coding will not dynamically update.

The solution to this problem is to move the labels to a different layer, and assign a neutral color to them. To do this, select **Tools / Element Properties** from the pull-down menus, and choose the **Labels** tab. Assign a new layer to the labels for all the elements, and check **Apply to Existing Objects**.

## A .6 How Do I Do a Profile Plot?

The **Profile Plot** window includes the selected upstream element and all elements linked in a direct downstream path to the outlet. There are two ways to open a **Profile Plot** window:

- Select **Tools / Profiling** from the pull-down menu to open the **Profiles Manager**. From here you can go through the **Profile Wizard**, by clicking the **Profile Management** button and selecting **Add**.

or

- Right-click an element, and select **Create Profile On…** from the pop-up menu. This will open up the **Profile Wizard** with all elements downstream of the selected element included in the list of elements to be profiled.

**The Options button allows you to control the display of detail layers in the Profile Window.**

## A .7 How Do I Change Units in a Column?

In a Table you may change the units of all the data within any column. To change the units:

1. Select **Use Local Units** from the **Options** menu in the **Tabular Report** dialog.
2. Right-click the column heading, or any data item within a column.
3. Select **Properties** from the pop-up menu.
4. Change the units and select **OK**. All data items in that column will change to the selected units.

**The change of units affects only the data in the Table. It DOES NOT change the units within your network design.**

# A .8 How Do I Access the Haestad Methods Knowledge Base?

You can access of hundreds of commonly asked questions at our online Knowledge Base.

The quickest way to access the Knowledge Base is to click the Globe Icon in the product toolbars. This will automatically log you on to our website. Simply click the **Knowledge Base** icon next to the Haestad product of interest.

If the computer you are using does not have internet access, you can log on to Knowledge Base at an alternate computer by going to "http://www.haestad.com" and entering the ClientCare portion of the website. You can then logon with the Product ID located in the back of the User's Manual or your PID number.

# A .9 How do I Model an Inverted Siphon (Depressed Sewer)?

There is no real trick to modeling an inverted siphon in SewerCAD. The inverted siphon consists of two or more gravity pipes (depending on changes of slope) that will be surcharged. Simply create pipes sloping downward and upward connected at a central junction. The gradually varied flow algorithm is robust enough to handle adverse slopes.

You can apply any bend losses to the central junction as a standard headloss. If you feel that friction losses are the predominant loss you can assume no headloss at the central junction. The central junction on the siphon (which may not be real) needs to be bolted or extend above the ground, so the hydraulic grade line is not reset if it exceeds the manhole's top.

If the siphon has multiple barrels in parallel, such that one takes the low flow and others come on line as flow increases, then it may be necessary to create a different physical alternatives for each possible number of pipes and replace the siphon by the equivalent pipe size in each alternative, or to use the diversion feature. See the *Gravity Flow Diversions Technical Supplement* for more information on setting up diversions and equivalent pipes.

Notes

# Appendix B

# SewerCAD Theory

## B .1 Overview

This appendix provides an overview of the methods that SewerCAD uses to compute flows and hydraulic grades throughout the system, including both gravity and pressure computations.

Some of the basic concepts underlying the calculations are as follows:

- SewerCAD can run both Steady State and Extended Period Analyses. Steady State Analyses model a single instant in time and are generally used to model a network under peak loading conditions. Extended Period Simulations model a network over a specified duration of time and can be used to model hydrograph loading, wet well capacities, and automated pump behavior.
- Loads are the sources of flow in the sanitary sewer system, and are categorized as sanitary (dry weather) loads, wet weather loads, and known loads. The total load at any given point may be a combination of these basic load types.
- Loads can be adjusted through the use of fixed or variable peaking factors during a Steady State analysis in order to analyze the system under a variety of conditions, such as daily average, minimum, and maximum scenarios. Common pre-defined variable peaking methods are included, but you may also specify your own as tables or equations.
- Loads can also be varied over time using loading patterns and hydrographs during an Extended Period Simulation.
- Gravity pipe headlosses are computed based on gradually varied flow profiles or approximate profiles. Either of these profile methods allows for free-surface (open channel) flow, full flow (as for a pipe that is submerged), and mixed conditions. Pressure pipe headlosses are based strictly on full-flow hydraulics.
- Gravity structure losses may be based on several common methodologies. Where appropriate, these calculations may account for pipe bend angles, structure benching, and other influential factors.
- During an Extended Period Simulation hydrographs are routed through the gravity pipes to account for translation and other effects.
- All or portions of gravity systems may be selected for automatic design. This preliminary design can be used to set pipe and structure elevations, as well as to size the pipes.

## B .2 Loading

SewerCAD classifies loads as sanitary (dry weather) loads, wet weather loads, and known loads.

Sanitary loads correspond to loads that result from human activity, and are not weather-dependent. Common sources of sanitary loads are various residential, commercial, recreational, and industrial usage.

Wet weather loads are related to rainfall activity, such as groundwater infiltration (water leaking into a pipe through cracks, joints, and other defects) and structure inflow (surface water entering a structure through the cover).

Known loads are typically used to model flows that have already been gathered from some other source, such as external calculations or field measurements.

In addition to these basic load types, there are also pumped loads. Pumped loads are special kinds of loads that represent sewage pumped into the gravity system via force mains. Pumped loads are optionally determined during calculations, and cannot be input directly.

## B.2.1 Common Load Types

There are two different loading types that can be applied as both wet weather and sanitary loads. The behavior of each loading type is the same regardless of how it is applied. The two common loads are:

- Hydrographs
- Pattern Loads

### Hydrographs

In SewerCAD you can enter time vs. flow data directly as a load. The hydrographs will then be directly added to any other loads coming to that point and then routed downstream

During a Steady State analysis a hydrograph loaded can be converted into a single load based on one of the following selected Steady State Loading options.

- **Peak** - The peak of the hydrograph will be used as the Steady State load.
- **Average** - The average of the hydrograph flows will be used as the Steady State load.
- **Minimum** - The minimum flow of the hydrograph will be used as the Steady State load.
- **Zero** - The hydrograph is disregarded during the Steady State run.

**The final flow of the hydrograph will remain constant for the duration of the simulation.**

**The Steady State Loading options are accessed by clicking the Options button on the Scenario / GO dialog.**

### Pattern Loads

A pattern load is comprised of a base load and an associated loading pattern. The pattern is a series of multipliers, which describes how the base load varies over time.

During a Steady State Analysis the entered base load is used as the load regardless of the applied loading pattern.

Extreme Flow Factors are not applied to entered base loads.

### Hydrographs vs. Pattern Loads

Hydrographs and Pattern loads are two distinct ways to describe how flow varies over time. Ultimately, you can attain the same results using either method but there are some behavioral and semantic differences that should be noted.

Pattern loads consists of a single average base load and a series of dimensionless multipliers used to delineate how the load varies over time. A hydrograph, simply, is a time-discharge series.

Hydrographs are usually applied as wet weather loads, and are generated using hydrologic methods, while patterns are more typically applied to sanitary loads. Patterns are developed based on predetermined variations in loading over the course of a day. The patterns are then assumed to represent templates for how the loads of a similar type vary over time. These statements represent typical usage of both loading types; they do not represent hard and fast rules.

During an Extended Period Simulation if the duration of the simulation exceeds the duration of a pattern then the pattern will repeat itself. If the duration of the simulation exceeds the duration of a hydrograph the last point of the hydrograph will remain constant for the extent of the remaining time.

## B.2.2 Sanitary (Dry Weather) Loading

The total sanitary load may be comprised of an unlimited number of individual sanitary loads. For example, the local load for a given manhole may be a combination of loads from an apartment building, a gas station, and a film development store, each with different loading characteristics. They can either be entered as:

- Unit Sanitary Loads
- Pattern Loads
- Hydrographs

Unit sanitary loads and pattern loads are calculated or entered as a base load, which represent the average loading on the system at that point. During a Steady State analysis the unit sanitary loads can be adjusted to represent peak or minimum loads using the Extreme Flow Factor methods. During an Extended Period Simulation a loading pattern can be applied to both the unit sanitary loads and pattern loads to described how the base load varies over time.

Hydrographs can also be applied as sanitary loads. They are meant to represent actual measured flow or hydrographs generated from other programs, and SewerCAD does not apply peaking factors or patterns to them.

### Unit Sanitary (Dry Weather) Loads

Unit sanitary (dry weather) loads are entered based on a number of contributing units, with a specified average load per unit, such as X-amount of flow per apartment resident. With each load type, you can then associate a peaking factor method or a loading pattern, allowing you to account for the knowledge that peaking factors and patterns are most likely different for residential and commercial areas.

Peaking factors are applied only during the Steady State analyses. During Extended Period Simulations loading patterns can be applied to the base loads generated from the unit sanitary loads to account for variations in sanitary inflow over time.

### Extreme Flow Factor

Sewer design and analysis generally considers a variety of loading conditions, such as minimum, average, and peak conditions. Base (average) sanitary loads are transformed into minimum or peak loads using an Extreme Flow Factor.

The most common type of Extreme Flow Factor (EFF) is the Variable Peaking Factor (PF).

$$Qpeaked = Qbase * EFF$$

| Where: | | | |
|---|---|---|---|
| | Qpeaked | = | Transformed flow (l/s, gpm) |
| | Qbase | = | Base flow (l/s, gpm) |
| | EFF | = | Extreme Flow Factor (unitless) |

**Extreme Flow Factor methods are only used during Steady State analyses. During Extended Period Simulations loading patterns can be applied to the Unit Dry Weather base loads.**

### Common Variable Peaking Factors

Some of the most common variable peaking factor (*PF*) calculation methods are:

*Babbitt*

$$PF = \frac{5.0}{\left(\frac{P}{1000}\right)^{0.20}}$$

Where: P = Contributing population (number of capita)

*Harmon*

$$PF = 1.0 + \frac{14.0}{4.0 + \left(\frac{P}{1000}\right)^{0.50}}$$

Where: P = Contributing population (number of capita)

*Ten States Standard (Great Lakes Upper Mississippi River Board)*

$$PF = \frac{18 + \sqrt{\frac{P}{1000}}}{4 + \sqrt{\frac{P}{1000}}}$$

Where: P = Contributing population (number of capita)

*Fedorov*

$$PF = \frac{2.69}{Q^{0.121}}$$

Where: Q = Base sanitary load (l/s)

**The Babbit peaking method does not converge to 1. For populations larger than 3,125,000 the peaking factor will become smaller than 1.**

## B.2.3 Wet Weather Loading

The Wet Weather Load represents the intrusion of rainfall water into the sewer system. Wet weather loads consist of groundwater infiltration, rainfall inflow, and illegal sump pump connections. Groundwater infiltration occurs in gravity pipes, while inflow occurs at manholes, pressure junctions, and wet wells.

Infiltration loads refer to wet weather loads entering pipes, where water leaks into the system through joints, cracks, and other defects. Inflow loads refer to wet weather loads entering structures, typically surface water entering through a structure's cover.

### Infiltration

Infiltration resulting from the presence of groundwater can be modeled for gravity pipes. It is combined with the loads at the upstream end of the pipe to determine the pipe's Total Wet Weather Flow.

There are several common methods of determining infiltration based on pipe characteristics, which is why SewerCAD allows infiltration to be defined with any of the following methods:

- **Proportional to Pipe Length** - The infiltration is specified as an Infiltration Rate per Unit of Pipe Length.
- **Proportional to Pipe Diameter-Length** - The infiltration is specified by an Infiltration Rate per Unit of Pipe Diameter times Pipe Length. The amount of infiltration is proportional to the pipe length and to the pipe diameter.
- **Proportional to Pipe Surface Area** - The infiltration is specified by an Infiltration Rate per Unit of Pipe Surface Area, where the pipe's surface area is calculated as its length multiplied by its full perimeter. The amount of infiltration is proportional to the pipe length and the pipe diameter.
- **Proportional to Count** - The infiltration is specified by count value, which may be the number of defects in the pipe, and the Infiltration Rate per Unit of Count.
- **Additional Infiltration** - Fixed amount of infiltration that is added to the total wet weather load. This value is constant regardless of the pipe's characteristics.
- **Hydrographs** - The infiltration is specified as a table of flow vs. time.
- **Pattern Loads -** The infiltration is specified as an average base load and a loading pattern.

During an Extended Period Simulation the five non-time-based methods will generate a single straight-line hydrograph producing a constant load for the duration of the simulation.

### Inflow

Inflow loads refer to wet weather loads entering structures, which are typically from surface water entering through a structure's cover, or pumped illegally into a force main system. Inflows can be entered as the following loading types:

- Hydrographs
- Pattern Loads

During Steady State analysis these loads can be modeled, and are combined with upstream wet weather loads to determine the Total Wet Weather Flow.

During Extended Period Simulations the loads are not classified once they enter the system they are added to together as a single lump hydrograph.

Inflows can be applied to manholes, wet wells, and pressure junctions.

## B.2.4 Known Loading

Known loads are a special type of fixed load. As with other fixed loads, known loads remain constant as they progress downstream and combine directly as a simple sum.

The special behavior of known loads occurs during a Steady State Analysis when another known load is specified at a downstream location. While most fixed loads combine directly under any circumstances, a non-zero known load at any location replaces all upstream known loads.

For this reason, known loads may be desirable for modeling loads that originate from external calculations or field measured data (loads that do not require SewerCAD to generate or sum them in any way).

During Extended Period Simulations, Known Flows are modeled as a single constant flow hydrograph over the duration of the simulation. They are added directly to the existing flows coming from upstream sources and are all lumped together as a single hydrograph for routing. **The Known Flows are additive and do not replace each other during Extended Period Simulations.**

# B .3 Gravity Pipe Hydraulics

## B.3.1 Basic Concepts

This documentation is intended to familiarize you with some of the methods used in this program's calculations. However, there is not a great deal of time spent on common hydraulic terms and equations, such as determination of wetted perimeter, hydraulic radius, hydraulic depth, and Reynolds number.

## B.3.2 Hydraulics and Energy Grades

### The Energy Principle

The first law of thermodynamics states that for any given system, the change in energy is equal to the difference between the heat transferred to the system and the work done by the system on its surroundings during a given time interval.

The energy referred to in this principle represents the total energy of the system minus the sum of the potential, kinetic, and internal (molecular) forms of energy, such as electrical and chemical energy. The internal energy changes are commonly disregarded in water distribution analysis because of their relatively small magnitude.

In hydraulic applications, energy is often represented as energy per unit weight, resulting in units of length. Using these length equivalents gives engineers a better feel for the resulting behavior of the system. When using these length equivalents, the state of the system is expressed in terms of head. The energy at any point within a hydraulic system is often represented in three parts:

- Pressure Head: $p/\gamma$
- Elevation Head: $z$
- Velocity Head: $V^2/2g$

Where:

| | | | |
|---|---|---|---|
| | $p$ | = | Pressure ($N/m^2$, $lb/ft^2$) |
| | $\gamma$ | = | Specific weight ($N/m^3$, $lb/ft^3$) |
| | $z$ | = | Elevation (m, ft) |
| | $V$ | = | Velocity (m/s, ft/s) |
| | $g$ | = | Gravitational acceleration constant ($m/s^2$, $ft/s^2$) |

These quantities can be used to express the headloss or head gain between two locations using the energy equation.

### The Energy Equation

In addition to pressure head, elevation head, and velocity head, there may also be head added to the system, by a pump for instance, and head removed from the system due to friction. These changes in head are referred to as head gains and headlosses, respectively. Balancing the energy across two points in the system, we then obtain the energy equation:

$$\frac{p_1}{g} + z_1 + \frac{V_1^{\,2}}{2g} + h_p = \frac{p_2}{g} + z_2 + \frac{V_2^{\,2}}{2g} + h_L$$

Where:
- p = Pressure (N/m$^2$, lb/ft$^2$)
- γ = Specific weight (N/m$^3$, lb/ft$^3$)
- z = Elevation at the centroid (m, ft)
- V = Velocity (m/s, ft/s)
- g = Gravitational acceleration constant (m/s$^2$, ft/s$^2$)
- hp = Head gain from a pump (m, ft)
- hL = Combined headloss (m, ft)

The components of the energy equation can be combined to express two useful quantities, which are the hydraulic grade and the energy grade.

### Hydraulic and Energy Grades

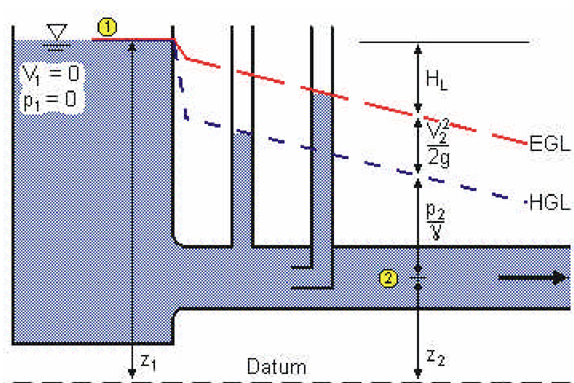
Hydraulic Grade

The hydraulic grade is the sum of the pressure head ( *p/γ* ) and elevation head ( *z* ). The hydraulic head represents the height to which a water column would rise in a piezometer. The plot of the hydraulic grade in a profile is often referred to as the hydraulic grade line, or HGL.

Energy Grade

The energy grade is the sum of the hydraulic grade and the velocity head ( $V^2/2g$ ). This is the height to which a column of water would rise in a pitot tube. The plot of the hydraulic grade in a profile is often referred to as the energy grade line, or EGL. At a lake or reservoir, where the velocity is essentially zero, the EGL is equal to the HGL, as can be seen in the following figure.

EGL and HGL




## B.3.3 Friction Loss Methods

There are many equations that approximate friction losses associated with the flow of liquid through a given section. Commonly used friction methods include:

- Chezy's Equation
- Kutter’s Equation
- Manning’s Equation

- Darcy-Weisbach Equation
- Colebrook-White Equation
- Hazen-Williams Equation

Friction losses are generally based on the relationships between fluid velocity, section roughness, depth of flow, and the friction slope (headloss per unit length of conduit).

## Chezy's Equation

Chezy's equation is rarely used directly, but it is the basis for several other methods, including Manning's equation and Kutter's equation. Chezy's equation is:

$$Q = C \cdot A \cdot \sqrt{R \cdot S}$$

Where:

| | | |
|---|---|---|
| Q | = | Discharge in the section ($m^3$/s, cfs) |
| C | = | Chezy's roughness coefficient ($m^{1/2}$/s, $ft^{1/2}$/s) |
| A | = | Flow area ($m^2$, $ft^2$) |
| R | = | Hydraulic radius (m, ft) |
| S | = | Friction slope (m/m, ft/ft) |

## Kutter's Equation

Kutter's equation can be used to determine the roughness coefficient in Chezy's formula, and is most commonly used for sanitary sewer analysis. Kutter's equation is as follows:

$$C = \frac{k_1 + \frac{k_2}{S} + \frac{k_3}{n}}{1 + \frac{n}{\sqrt{R}} \cdot \left( k_1 + \frac{k_2}{S} \right)}$$

Where:

| | | |
|---|---|---|
| C | = | Chezy's roughness coefficient($m^{1/2}$/s, $ft^{1/2}$/s) |
| S | = | Friction slope (m/m, ft/ft) |
| R | = | Hydraulic radius (m, ft) |
| n | = | Kutter's roughness (unitless) |
| $k_1$ | = | Constant (23.0 for SI, 41.65 for US) |
| $k_2$ | = | Constant (0.00155 for SI, 0.00281 for US) |
| $k_3$ | = | Constant (1.0 for SI, 1.811 for US) |

**Kutter's roughness coefficients are the same as Manning's roughness coefficients.**

## Manning's Equation

Manning's equation is one of the most popular methods in use today for free surface flow (and, like Kutter's equation, is based on Chezy's equation). For Manning's equation, the roughness coefficient in Chezy's equation is calculated as:

$$C = k \cdot \frac{R^{1/6}}{n}$$

Where: C = Chezy's roughness coefficient ($m^{1/2}$/s, $ft^{1/2}$/s)

R = Hydraulic radius (m, ft)
n = Manning's roughness ($s/m^{1/3}$)
k = Constant (1.00 $m^{1/3}/m^{/13}$, 1.49 $ft^{1/3}/m^{1/3}$)

Substituting this roughness into Chezy's equation, we obtain the well-known Manning's equation:

$$Q = \frac{k}{n} \cdot A \cdot R^{2/3} \cdot S^{1/2}$$

Where:
Q = Discharge ($m^3/s$, cfs)
k = Constant(1.00 $m^{1/3}/m^{/13}$, 1.49 $ft^{1/3}/m^{1/3}$)
n = Manning's roughness ($s/m^{1/3}$)
A = Flow area ($m^2$, $ft^2$)
R = Hydraulic radius (m, ft)
S = Friction slope (m/m, ft/ft)

**Manning's roughness coefficients are the same as the roughness coefficients used in Kutter's equation.**

## Darcy-Weisbach Equation

Because of non-empirical origins, the Darcy-Weisbach equation is viewed by many engineers as the most accurate method for modeling friction losses. It most commonly takes the following form:

$$h_f = f \cdot \frac{L}{D} \frac{V^2}{2g}$$

Where:
hf = Headloss (m, ft)
f = Darcy-Weisbach friction factor (unitless)
D = Pipe diameter (m, ft)
L = Pipe length (m, ft)
V = Flow velocity (m/s, ft/s)
g = Gravitational acceleration constant ($m/s^2$, $ft/s^2$)

For section geometries that are not circular, this equation is adapted by relating a circular section's full-flow hydraulic radius to its diameter:

$$D = 4R$$

Where:
R = Hydraulic radius (m, ft)
D = Diameter (m, ft)

This can then be rearranged to the form:

$$Q = A \cdot \sqrt{8g \cdot \frac{R \cdot S}{f}}$$

Where:
- Q = Discharge ($m^3/s$, cfs)
- A = Flow area ($m^2$, $ft^2$)
- R = Hydraulic radius (m, ft)
- S = Friction slope (m/m, ft/ft)
- f = Darcy-Weisbach friction factor (unitless)
- g = Gravitational acceleration constant ($m/s^2$, $ft/s^2$)

The Swamme and Jain equation can then be used to calculate the friction factor.

*Swamme and Jain Equation:*

$$f = \frac{1.325}{\left[\ln\left(\frac{k}{3.7D} + \frac{5.74}{R_e^{0.9}}\right)\right]^2}$$

Where:
- f = Friction factor (unitless)
- k = Roughness height (m, ft)
- D = Pipe diameter (m, ft)
- Re = Reynolds number (unitless)

The friction factor is dependent on the Reynolds number of the flow, which is dependent on the flow velocity, which is dependent on the discharge. As you can see, this process requires the iterative selection of a friction factor until the calculated discharge agrees with the chosen friction factor.

**The Kinematic Viscosity is used in determining the friction coefficient in the Darcy-Weisbach Friction Method. The default units are initially set by Haestad Methods.**

## Colebrook-White Equation

The Colebrook-White equation is used to iteratively calculate for the Darcy-Weisbach friction factor:

Free Surface

$$\frac{1}{\sqrt{f}} = -2\log\left(\frac{k}{14.8R} + \frac{2.51}{R_e\sqrt{f}}\right)$$

**Full Flow (Closed Conduit)**

$$\frac{1}{\sqrt{f}} = -2\log\left(\frac{k}{12.0R} + \frac{2.51}{R_e\sqrt{f}}\right)$$

Where:
- Re = Reynolds Number (unitless)
- k = Darcy-Weisbach roughness height (m, ft)
- f = Friction factor (unitless)
- R = Hydraulic radius (m, ft)

### Hazen-Williams Equation

The Hazen-Williams Formula is frequently used in the analysis of pressure pipe systems (such as water distribution networks and sewer force mains). The formula is as follows:

$$Q = k \cdot C \cdot A \cdot R^{0.63} \cdot S^{0.54}$$

Where:
- Q = Discharge in the section ($m^3/s$, cfs)
- C = Hazen-Williams roughness coefficient (unitless)
- A = Flow area ($m^2$, $ft^2$)
- R = Hydraulic radius (m, ft)
- S = Friction slope (m/m, ft/ft)
- k = Constant (0.85 for SI, 1.32 for US).

## B.3.4 Flow Regime

The hydraulic grade in a flow section depends heavily on the tailwater conditions, pipe slope, discharge, and other conditions. The basic flow regimes that a pipe may experience include:

- Pressure Flow
- Uniform (Normal) Flow
- Critical Flow
- Subcritical Flow
- Supercritical Flow

**Based on the gradually varied flow analysis, different portions of any given pipe may be under different flow regimes.**

### Pressure Flow

When a pipe is surcharged, headlosses are simply based on the full barrel area and wetted perimeter. Because these characteristics are all functions of the section shape and size, friction loss calculations are greatly simplified by pressurized conditions.

### Uniform Flow and Normal Depth

Uniform flow refers to a hydraulic condition where the discharge and cross-sectional area, and therefore the velocity, are constant throughout the length of the channel or pipe. For a pipe flowing full, all that this requires is that the pipe be straight and have no contractions or expansions. For a non-full section, however, there are a few additional points of interest:

- In order for the cross-sectional area to remain the same, the depth of flow must be constant throughout the length of the channel. This requires that the friction slope equal the constructed slope. This depth is called normal depth.
- Since the hydraulic grade line parallels the invert of the section and the velocity does not change, the energy grade line is parallel to both the hydraulic grade line and the section invert under uniform flow conditions.

In prismatic channels, flow conditions will typically approach normal depth if the channel is sufficiently long.

### Critical Flow, Critical Depth, and Critical Slope

Critical flow occurs when the specific energy of the section is at a minimum. This condition is defined by the situation where:

$$\frac{A^3}{T} = \frac{Q^2}{g}$$

Where:
- $A$ = Area of flow ($m^2$, $ft^2$)
- $T$ = Top width of flow (m, ft)
- $Q$ = Section discharge ($m^3/s$, $ft^3/s$)
- $g$ = Gravitational acceleration ($m/s^2$, $ft/s^2$)

This is a relatively simple computation for simple geometric shapes, but can require iterative calculation for more complex shapes (such as arches). Some sections may even have several valid critical depths, making numerical convergence more difficult.

Critical depth refers to the depth of water in a channel for which the specific energy is at its minimum.

Critical slope refers to the slope at which the critical depth of a pipe would be equal to the normal depth.

#### *Subcritical Flow*

Subcritical flow refers to any flow condition where the Froude number is less than 1.0. For this condition, the depth is above critical depth, and the velocity is below the critical depth velocity.

#### *Supercritical Flow*

Supercritical flow refers to any condition where the Froude number, or the ratio of internal forces to gravity forces, is greater than 1.0. For this condition, the depth is below critical depth, and the velocity is above the critical depth velocity.

## B.3.5 Gradually Varied Flow Analysis

For free surface flow, depth rarely remains the same throughout the length of a channel or pipe. Starting from a boundary control depth, the depth changes gradually, increasing or decreasing until normal depth is achieved (if the conduit is sufficiently long). The determination of a boundary control depth depends on both the tailwater condition and the hydraulic characteristics of the conduit. The areas of classification for gradually varied flow analysis are:

- Slope Classification
- Zone Classification
- Profile Classification

### Slope Classification

The constructed slope of a conduit is a very important factor in determining the type of gradually varied flow profile that exists. Slopes fall into one of five types, all of which are handled by the program:

- Adverse Slope
- Horizontal Slope
- Hydraulically Mild Slope
- Critical Slope
- Hydraulically Steep Slope

Any pipe can qualify as only one of these slope types for a given discharge. For differing flows, though, a pipe may change between qualifying as a mild, critical, and steep slope. These slopes do not relate to just the constructed slope, but to the constructed slope relative to the critical slope for the given discharge.

### *Adverse Slope*

Adverse slope occurs when the upstream invert elevation of a pipe is actually below the downstream invert elevation. Normal depth is undefined for adverse slopes, since no amount of positive flow would result in a rising friction slope. Most flow conditions for adverse sloping pipes are subcritical.

Pipes are typically not designed to be adverse, so most situations with adverse slopes are due to construction errors or other unusual circumstances. Adverse pipes may cause some concern beyond the hydraulic capacity of the system, because stagnant water, excessive clogging, and other non-desirable conditions may result.

### *Horizontal Slope*

As the name suggests, a horizontal slope results when a pipe’s upstream and downstream invert elevations are the same. Normal depth for a horizontal pipe is theoretically infinite, although critical depth may still be computed. Like adverse slopes, most flow conditions for horizontal pipes are subcritical.

### *Hydraulically Mild Slope*

A hydraulically mild slope is a condition where the constructed slope is less than the critical slope. For this condition, the section’s normal depth is above critical depth, and the flow regime is usually subcritical.

### *Critical slope*

A pipe or channel may have exactly the same slope as the critical slope for the discharge it carries. This is a very uncommon occurrence, but it is possible and the program does calculate it appropriately. Critical depth is an inherently unstable surface, so flow is most likely to be subcritical for these slopes.

### *Hydraulically Steep Slope*

A hydraulically steep slope is a condition where the constructed slope is greater than the critical slope. For this condition, the section’s normal depth is below critical depth, and the flow regime is usually supercritical. However, high tailwater conditions may cause flow to be subcritical.

## Zone Classification

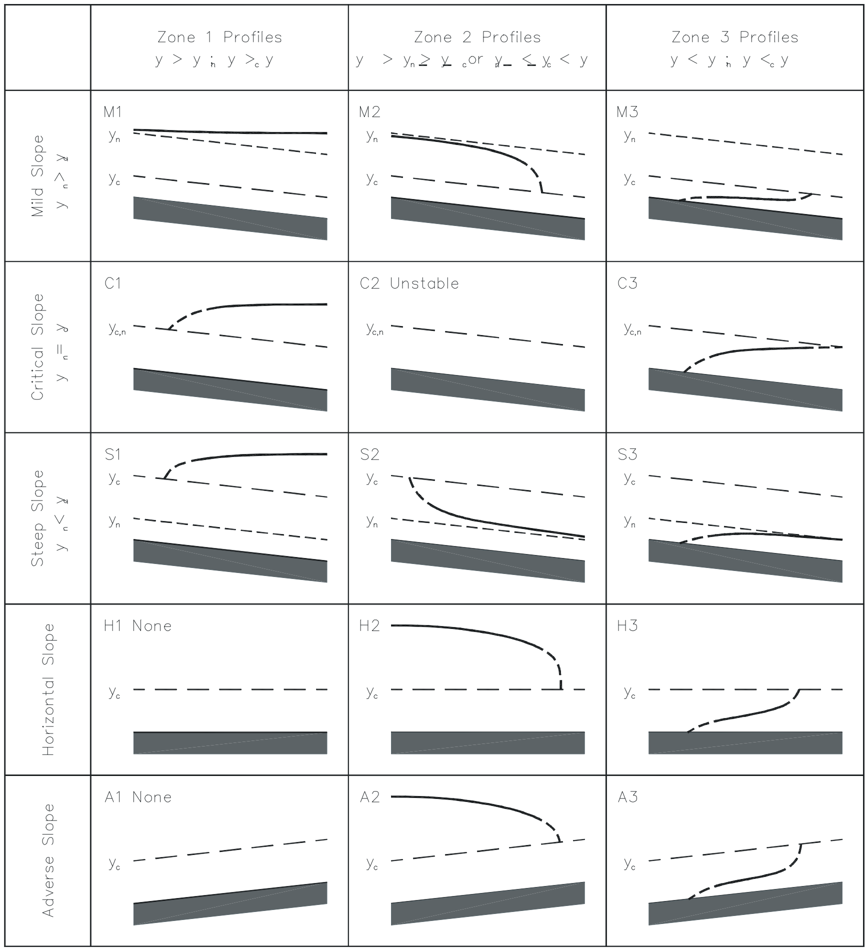
There are three zones that are typically used to classify gradually varied flow:

- Zone 1 is where actual flow depth is above both normal depth and critical depth.
- Zone 2 is where actual flow depth is between normal depth and critical depth.
- Zone 3 is where actual flow depth is below both normal depth and critical depth.

## Profile Classification

The gradually varied flow profile classification is simply a combination of the slope classification and the zone classification. For example, a pipe with a hydraulically mild slope and flow in zone 1 would be considered a Mild-1 profile (M1 for short). The program will analyze most profile types, but will not analyze certain flow profile types that occur rarely in conventional sewer system such as H3, M3, and S3.

Profile Classification




## B.3.6 Energy Balance

Even for gradually varied flow, the solution is still a matter of balancing the energy between the two ends of a pipe segment. The energy equation as it relates to each end of a segment is as follows (note that the pressures for both ends are zero, since it is free surface flow):

$$Z_1 + \frac{V_1^2}{2g} = Z_2 + \frac{V_2^2}{2g} + H_L$$

Where:

| | | |
|---|---|---|
| $Z_1$ | = | Hydraulic grade at upstream end of the segment (m, ft) |
| $V_1$ | = | Velocity at the upstream end (m/s, ft/s) |
| $Z_2$ | = | Hydraulic grade at the downstream end of the segment (m, ft) |
| $V_2$ | = | Velocity at the downstream end (m/s, ft/s) |
| $H_L$ | = | Loss due to friction - other losses are assumed to be zero (m, ft) |
| g | = | Gravitational acceleration constant ($m/s^2$, $ft/s^2$) |

The friction loss is computed based on the average rate of friction loss along the segment and the length of the segment. This relationship is as follows:

$$H_L = S_{Avg} \cdot \Delta x = \frac{S_1 + S_2}{2} \Delta x$$

Where:
- $H_L$ = Loss across the segment (m, ft)
- $S_{avg}$ = Average friction slope (m/m, ft/ft)
- $S_1$ = Friction slope at the upstream end of the segment (m/m, ft/ft)
- $S_2$ = Friction slope at the downstream end of the segment (m/m, ft/ft)
- $\Delta x$ = Length of the segment being analyzed (m, ft)

The conditions at one end of the segment are known through assumption or from a previous calculation step. Since the friction slope is a function of velocity, which is a function of depth, the depth at the other end of the segment can be found through iteration. There are two primary methods for this iterative solution, the Standard Step method and the Direct Step method.

### Standard Step Method

The standard step method of gradually varied flow energy balance involves dividing the channel into segments of known length and solving for the unknown depth at one end of the segment, starting with a known or assumed depth at the other end. The standard step method is the most popular method of determining the flow profile because it can be applied to any channel, not just prismatic channels.

### Direct Step Method

The direct step method is based on the same basic energy principles as the standard step method, but takes a slightly different approach towards the solution. Instead of assuming a segment length and solving for the depth at the end of the segment, the direct step method assumes a depth and then solves for the segment length.

Because it generates better resolution within the changing part of the profile, the gravity flow algorithm of StormCAD and SewerCAD primarily use the direct step method to compute gradually varied flow profiles.

## B.3.7 Mixed Flow Profiles

Although the hydraulic slope of a pipe will be the same throughout its length, a pipe may contain several different profile types. The transitions that may be encountered include:

- Sealing (Surcharging) Conditions
- Rapidly Varied Flow (Hydraulic Jumps)

### Sealing (Surcharging) Conditions

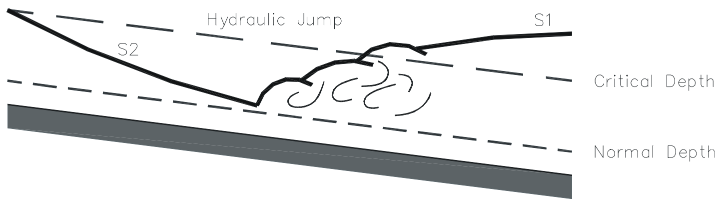
There may be conditions such that part of the section is flowing full, while part of the flow remains open. These conditions are called sealing conditions, and the sections are analyzed in separate parts. For sealing conditions, the portion of the section flowing full is analyzed as pressure flow, and the remaining portion is analyzed with gradually varied flow techniques.

### Rapidly Varied Flow

Rapidly varied flow is turbulent flow resulting from the abrupt and pronounced curvature of flow streamlines into or out of a hydraulic control structure. Examples of rapidly varied flow include hydraulic jumps, bends, and bridge contractions.

The hydraulic phenomenon that occurs when the flow passes rapidly from supercritical to subcritical flow is called a hydraulic jump. The most common occurrence of this within a gravity flow network occurs when there is a steep pipe discharging into a particularly high tailwater, as shown in the following figure.

Hydraulic Jump




There are significant losses associated with hydraulic jumps, due to the amount of mixing and hydraulic turbulence that occurs. These forces are also highly erosive, so engineers typically try to prevent jumps from occurring in gravity flow systems, or at least try to predict the location of these jumps in order to provide adequate channel, pipe, or structure protection. The program does not perform any specific force analyses that seek to precisely locate the hydraulic jump, nor does it identify the occurrence of jumps that might happen as flows leave a steep pipe and enter a mild pipe. Rather it performs analyses sufficient to compute grades at structures.

## B.3.8 Backwater Analysis

The classic solution of gravity flow hydraulics is via a backwater analysis. This type of analysis starts at the network outlet under free discharge, submerged, or tailwater control, and proceeds in an upstream direction.

Steep pipes tend to "interrupt" the backwater analysis, and reset the hydraulic control to critical depth at the upstream end of the steep pipe. A frontwater analysis may be needed for a steep profile (such as an S2), with the backwater analysis recommencing from the upstream structure.

### Free Outfall

This program lets you define the tailwater condition at the outlet as either Free Outfall, Crown Elevation or User-Specified.

For a pipe with a hydraulically steep slope, the Free Outfall condition will yield a starting depth equal to normal depth in the pipe. For a pipe with a hydraulically mild slope, the Free Outfall condition will yield a starting depth equal to critical depth. When an outlet has multiple incoming pipes, the Free Outfall condition yields a starting elevation equal to the lowest of the individual computed elevations.

The Crown condition should be used when the pipe discharges to an outlet where the water surface elevation is equal to the elevation of the top of the pipe.

### Structure Flooding

Flooding at manholes in SewerCAD and inlets in StormCAD occurs whenever the elevation of water is above the structure rim elevation. When this occurs, the backwater analysis will continue by resetting the hydraulic grade to the structure rim elevation or ground elevation, whichever is higher. However, if a structure is defined with a bolted cover, the hydraulic grade is not reset to the rim elevation.

In actual flooding situations, flows may be diverted away from the junction structure and out of the system, or attenuated due to surcharged storage. In this program, even though the governing downstream boundary for the next conduit is artificially lowered to prevent the propagation of an incorrect backwater, the peak discharges at the structure are conserved and are not reduced by the occurrence of flooding at a junction.

## B.3.9 Frontwater Analysis

The program will perform a frontwater analysis in a steep pipe operating under supercritical flow, since these pipes are typically entrance controlled. The hydraulic control is at the upstream end of the conduit, and the gradually varied flow analysis will proceed in a downstream direction until either the normal depth is achieved, a hydraulic jump occurs, or the end of the pipe is encountered.

**The program's algorithm is fundamentally based on backwater analysis. As a result, a continuous frontwater analysis is not performed through two or more consecutive steep pipes. This is a performance trade-off that has little impact in evaluating performance of the collection system in most situations. The assumption of critical depth at the upstream end results in a conservative depth in all cases, and is exactly correct at the point of the steep run furthest upstream.**

## B.3.10 Pipe Average Velocity

### Average Velocity Methods

Several common methods for computing a pipe's average velocity are available:

- Uniform Flow Velocity
- Full Flow Velocity
- Simple Average Velocity
- Weighted Average Velocity

### Uniform Flow Velocity

The uniform flow velocity of a pipe is obtained by calculating the velocity in the pipe at normal depth. If the normal depth corresponds to a surcharged condition, the full flow velocity is used instead.

### Full Flow Velocity

The full flow velocity corresponds to the velocity when the pipe is flowing full. The flow area is equal to the entire cross-sectional area of the pipe.

### Simple Average Velocity

The simple average velocity is computed by:

$$V_a = \frac{V_u + V_d}{2}$$

Where:
$V_a$ = Average velocity (m/s, ft/s)
$V_U$ = Upstream velocity (m/s, ft/s)
$V_d$ = Downstream velocity (m/s, ft/s)

**The Simple Average Velocity method does not account for any depth changes between the two ends of the pipe as the weighted average velocity method does.**

### Weighted Average Velocity

To compute the weighted average velocity, the simple average velocity of each profile segment is considered and given a weight based on its length:

$$V_a = \sum_{i=1}^{n} \left( \frac{V_{ui} + V_{di}}{2} \right) \cdot \left( \frac{L_i}{L_t} \right)$$

Where: $V_a$ = Average velocity for the pipe (m/s, ft/s)
$V_{ui}$ = Upstream velocity for segment i (m/s, ft/s)
$V_{di}$ = Downstream velocity for segment i (m/s, ft/s)
$L_i$ = Length of the profile segment i (m, ft)
$L_t$ = Total length of the pipe (m, ft)

### Pipe Average Velocity and Travel Time

The travel time though each pipe is computed as:

$$t = L / V$$

Where: t = Time of travel through the pipe (s)
V = Average velocity though the pipe (m/s, ft/s)
L = Length of the pipe (m, ft)

## B.3.11 Capacity Analysis (Approximate Profiles)

Traditionally, gravity pipe analyses and designs have not included the calculation-intense process of estimating a gradually varied flow profile. With this program, you have the option of determining discharge using gradually varied flow, or using the more traditional Capacity Analysis option. Capacity analysis still uses a backwater approach, with the profile type for a pipe being primarily dependent on the pipe's full flow capacity and downstream hydraulic grade.

The capacity analysis is advantageous over the gradually varied flow analysis in terms of processing time. If you are dealing with a relatively large network and you wish to arrive quickly at reasonable approximation then the capacity analysis is the way to go. The gradually varied flow algorithms are more rigorous and generate solutions that more closely reflect reality.

There are two basic approximate profile cases: the Full Capacity Profile and the Excess Capacity Profile.

#### *Full Capacity Profiles*

Full capacity profiles occur when the pipe's actual discharge is greater than or equal to the pipe's full flow capacity. In these cases, the downstream depth is taken as the greater of the actual downstream hydraulic grade or the free discharge tailwater elevation. The free discharge tailwater depth is commonly approximated as halfway between the crown of the pipe and the pipe's critical depth (in accordance with the U.S. Federal Highway Administration's HDS-5).

Starting from the tailwater elevation, the pipe's full flow friction slope is used to determine the hydraulic grade at the upstream end of the profile.

#### *Excess Capacity Profiles*

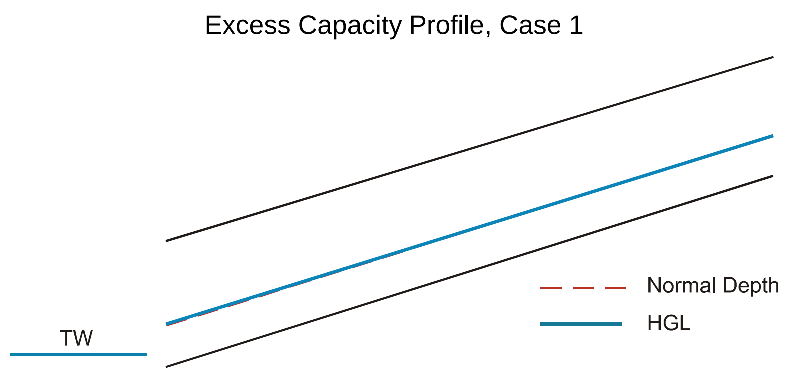
Excess capacity profiles occur when the full flow capacity of the pipe is greater than the actual flow in the pipe. For these profiles, there are three basic tailwater conditions:

**Case 1** - Hydraulic grade downstream less than or equal to normal depth.

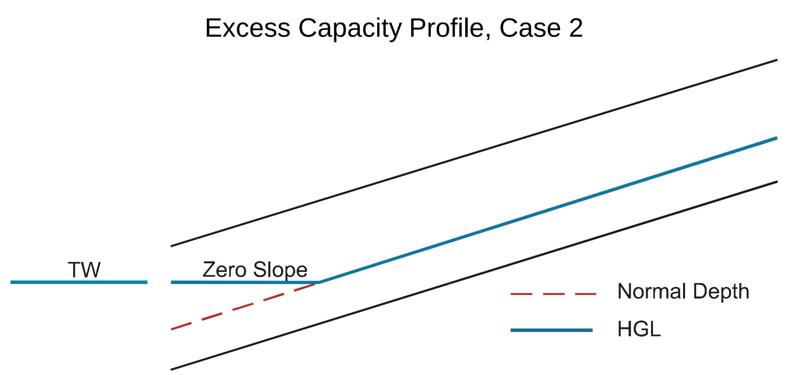
**Case 2** - Hydraulic grade downstream greater than normal depth, and less than or equal to pipe crown.

**Case 3** - Hydraulic grade downstream greater than or equal to pipe crown.

Excess Capacity Profile, Case 1 (Hydraulic Grade <= Normal Depth):
If the downstream depth in the pipe is at or below the pipe's normal depth, normal depth is assumed for the pipe's entire length.

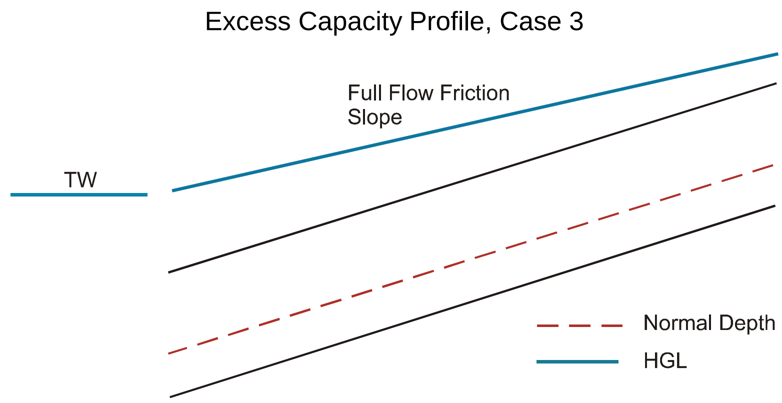



Excess Capacity Profile, Case 2 (Normal Depth < Hydraulic Grade <= Pipe Crown)
When the hydraulic grade is above the pipe's normal depth but below the top of the pipe, a friction slope of zero is assumed until it either intersects the pipe's normal depth or reaches the end of the pipe.



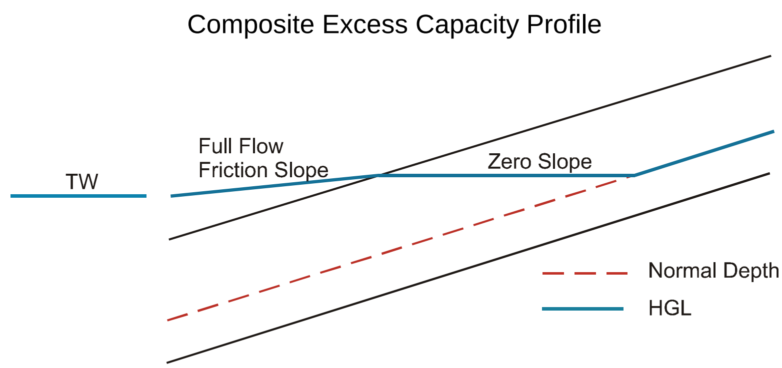

Excess Capacity Profile, Case 3 (Hydraulic Grade >= Pipe Crown)
If the hydraulic grade is above the pipe crown, the hydraulic grade continues upstream following the pipe's full flow friction slope. This slope will continue until it either intersects the pipe crown or reaches the end of the pipe.

**If the full friction slope intersects the crown of the pipe, the profile will continue with a Case 2 profile analysis.**




### Composite Excess Capacity Profiles

An excess capacity profile may actually be a composite of two more simple profiles. Consider the case below, where the tailwater is above the crown of the pipe. In this case, the profile begins as a Case 3 profile. Where the full flow friction slope intersects the crown of the pipe, the profile changes to a Case 2 profile, following a flat slope until it reaches normal depth. Where normal depth is intersected, a Case 1 profile begins, extending all the way to the upstream end of the pipe.




# B .4 Junction Headlosses and Minor Losses

## B.4.1 Junction Headlosses

### Structure Headloss

When water flows through a junction structure, there are headlosses associated with mixing, change of direction, and so forth. This section deals with the computation of these losses based on the following popular methods:

- Absolute
- Standard
- HEC-22 Energy
- AASHTO
- Generic

Structure headlosses are used to determine the hydraulic grade to use as the tailwater condition for upstream pipes during the backwater analysis. With the exception of the HEC-22 Energy method, the headloss through the structure is assumed to be the same for each incoming pipe.

## Headloss - Absolute Method

The absolute method is the simplest of the headloss methods. The structure headloss becomes an editable value, which is then used during calculations. No computations relating to velocity, confluence angle, or other factors are needed.

## Headloss - Standard Method

The standard method calculates structure headloss based on the exit pipe's velocity. The exit velocity head is multiplied by a user-entered coefficient to determine the loss:

$$h_s = K \cdot \frac{V_o^2}{2g}$$

Where:

- $h_S$ = Structure headloss (ft, m)
- $V_O$ = Exit pipe velocity (ft/s, m/s)
- $g$ = Gravitational acceleration constant (ft/s$^2$, m/s$^2$)
- $K$ = Headloss coefficient (unitless)

For suggested coefficient values for various structure configurations, see the Typical Headloss Coefficient table at the end of this chapter.

## Headloss - Generic Method

The generic method computes the structure headloss by multiplying the velocity head of the exit pipe by the user-entered downstream coefficient and then subtracting the velocity head of the governing upstream pipe multiplied by the user-entered upstream coefficient.

$$h_s = K_0 \cdot \frac{V_o^2}{2g} - K_1 \cdot \frac{V_1^2}{2g}$$

Where:

- $h_S$ = Structure headloss (ft, m)
- $V_O$ = Exit pipe velocity (ft/s, m/s)
- $K_O$ = Downstream coefficient (unitless)
- $V_1$ = Governing upstream pipe velocity (ft/s, m/s)
- $K_1$ = Upstream coefficient (unitless)
- $g$ = Gravitational acceleration constant (ft/s$^2$, m/s$^2$)

If there are multiple upstream pipes entering the junction then the program must choose one of the pipes to use in the calculation. The pipe that is chosen is considered the governing upstream pipe. The governing upstream pipe is selected based on one of the following methodologies:

- The upstream pipe with the maximum flow times velocity
- The upstream pipe with the maximum velocity head
- The upstream pipe with the minimum bend angle

The default method for selecting the governing upstream pipe is to choose the pipe with the maximum flow times velocity. However, the user can select one of the other options through the generic structure loss options.

## Headloss-HEC-22 Energy Method

Similar to the standard method, the HEC-22 Energy method (from the FHWA's *Urban Drainage Design Manual, Hydraulic Engineering Circular No. 22*) correlates structure headloss to the velocity head in the outlet pipe using a coefficient. Experimental studies have determined that this coefficient can be approximated by:

$$K = K_o C_D C_d C_Q C_p C_B$$

Where:
- $K$ = Adjusted headloss coefficient
- $K_O$ = Initial headloss coefficient based on relative junction size
- $C_D$ = Correction factor for the pipe diameter
- $C_d$ = Correction factor for flow depth
- $C_Q$ = Correction for relative flow
- $C_P$ = Correction for plunging flow
- $C_B$ = Correction factor for benching

### *Special Assumptions*

The HEC-22 Energy method documentation is written with a limited range of applicability. Many of the equations are written on the basis of pipe diameter, structure diameter, and so on. Since StormCAD and SewerCAD offer non-circular pipes and non-circular structures, this creates the need for some interpretation of the term "diameter."

In some cases, the intent of the methodology is to compare the size of one pipe to another pipe, or to the size of a structure. In these cases an equivalent diameter is used, which is computed from the full area of the pipe or structure. Equivalent diameter is the diameter of a circle with the area equal to the area of the examined pipe or structure.

In other cases, the intent of the methodology is to compare depths within the structure. For these cases, the rise (height) of the pipes is used in place of "diameter."

### *Pressure Flow, Free Surface Flow, and Transitional Flow*

Throughout the documentation for HEC-22 Energy losses, you will see references to "pressure flow", "free surface flow", and "transitional flow".

Pressure flow (submerged flow) is assumed to be any condition for which the depth of water above the outlet pipe invert is greater than 3.2 times the height of the outlet pipe.

Free surface flow (unsubmerged flow) is assumed to be any condition for which the depth of water above the outlet pipe invert is less than the height of the pipe.

Transitional flow is any condition between pressure flow and free surface flow.

### *Initial Headloss Coefficient*

The initial headloss coefficient, which is based on relative junction size, is calculated as:

$$K_o = 0.1\left(\frac{b}{D_e}\right)(1-\sin\theta)+1.4\left(\frac{b}{D_e}\right)^{0.15}\sin\theta$$

Where:
- $\theta$ = Deflection angle between inflow and outflow pipes
- $b$ = Equivalent diameter of the structure (m, ft)
- $D_e$ = Equivalent diameter of the outlet pipe (m, ft)

**The angle used in this equation is a deflection angle, so a straight run has a deflection angle of 180°. The bend angle in this case is 0°.**

### *Correction for Pipe Diameter*

The correction factor due to differences in pipe size is calculated only for pressure flow situations. For non-pressure situations, a value of 1.0 is used.

$$C_D = \left(\frac{D_o}{D_i}\right)^3$$ for pressure flow; $C_D$ = 1.0 for non-pressure flow

Where: $D_O$ = Outlet pipe rise (m, ft)

$D_i$ = Inflow pipe rise (m, ft)

### *Correction for Flow Depth*

The correction factor for flow depth is used only in cases of free surface flow or transitional flow. For pressure flow, a value of 1.0 is used.

$$C_d = \left(\frac{d_{aho}}{D_e}\right)^{0.6}$$ for pressure flow; $C_d$ = 1.0 for pressure flow

Where $d_{aho}$ = Water depth in the structure (m, ft)

$D_e$ = Outlet pipe rise (m, ft)

### *Correction for Relative Flow*

The correction factor for relative flow is calculated only when the invert elevation for the pipe in question is approximately equal to the invert elevation of the outlet pipe and at least one other pipe. Otherwise, a value of 1.0 is used.

$$C_Q = (1 - 2\sin\theta)\left(1 - \frac{Q_i}{Q_o}\right)^{0.75} + 1$$

Where: $\theta$ = Deflection angle between inflow and outflow pipes

$Q_i$ = Flow in the inflow pipe ($m^3/s$, cfs)

$Q_o$ = Flow in the outflow pipe ($m^3/s$, cfs)

**The term "approximately equal" is quite a vague definition for when to use relative flow corrections. StormCAD and SewerCAD enable you to change the tolerance for "approximately equal" elevations so that you can use your judgment to fine-tune the HEC-22 methodology.**

### *Correction for Plunging Flow*

The correction factor for plunging flow accounts for the effect that flow plunging into a junction from another inflow pipe has on the inflow pipe for which the headloss is calculated. It is calculated only when vertical distance from the invert of the plunge pipe to the center of the outflow pipe is greater than the depth in the structure relative to the outlet pipe invert. Otherwise a value of 1.0 is used.

$$C_p = 1 + 0.2\left(\frac{h}{D_o}\right)\left(\frac{h - d_{aho}}{D_o}\right)$$

Where:

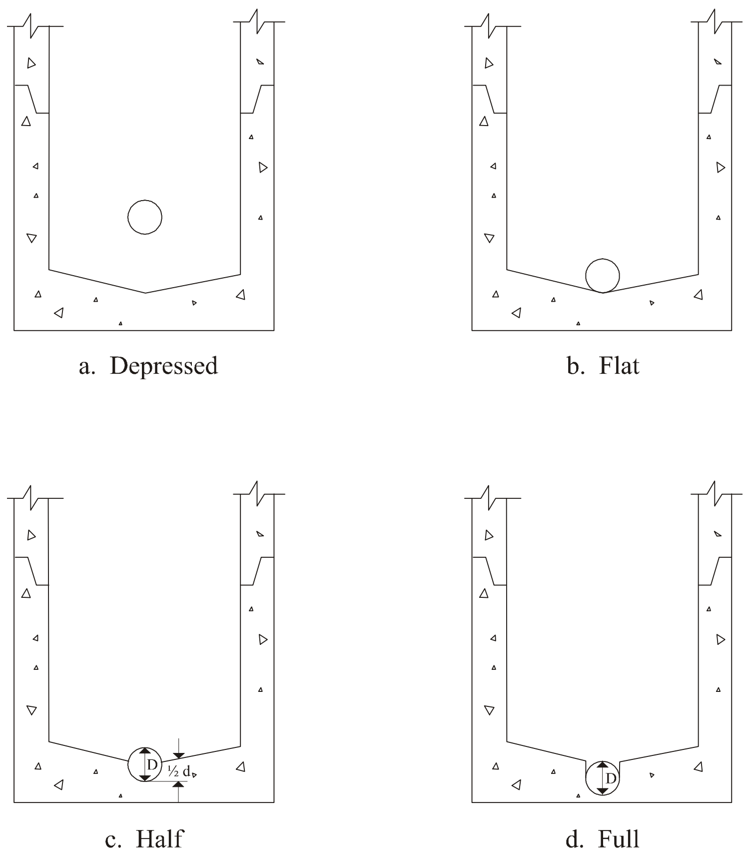
- $h$ = Vertical distance from invert of the plunge pipe to the center of the outflow pipe (m, ft)
- $D_e$ = Outflow pipe rise (m, ft)
- $d_{aho}$ = Water depth in the junction relative to the outflow pipe invert (m, ft)

### *Correction for Benching*

The correction factor for structure benching is similar to the shaping correction factor used in the AASHTO structure loss method. The correction accounts for smoother transitions from the inflow pipe to the outflow pipe based on the presence (or lack) of shaping in the bottom of the structure.

The following figure represents the four types of benching:

Types of Benching




By default, the program uses the values documented in HEC-22 (and presented in the following table) for pressure and free surface flow, but the user can change these values. For transitional flow, the program interpolates from the table linearly, based on the actual ratio of depth in the access hole to the height of the outflow pipe.

| Bench Type | Correction Factor, *CB* | |
|---|---|---|
| | **Pressure *** | **Free Surface **** |
| Flat Floor | 1.00 | 1.00 |
| Depressed Floor | 1.00 | 1.00 |
| Half Bench | 0.95 | 0.15 |
| Full Bench | 0.75 | 0.07 |

* pressure flow, $d_{aho} / D_e > 3.2$

** free surface flow, $d_{aho} / D_e < 1.0$

($d_{aho}$ is the water depth in the structure above the outlet pipe invert and $D_O$ is the outlet pipe diameter. )

## Headloss-AASHTO Method

### *Headloss - AASHTO Method*

The AASHTO method (as defined in the AASHTO *Model Drainage Manual*) for structure headloss is based on power-loss methodologies. This method can be summarized by the following equation:

$$h_s = (h_c + h_b + h_e) \cdot C_n \cdot C_s$$

Where:
- $h_S$ = Structure headloss (m, ft)
- $h_C$ = Contraction loss (m, ft)
- $h_b$ = Bend loss (m, ft)
- $h_e$ = Expansion loss (m, ft)
- $C_n$ = Correction factor for non-piped flow (unitless)
- $C_S$ = Correction factor for shaping (unitless).

### *AASHTO Contraction Loss*

The contraction loss is due to flow transitioning from large-area, low-velocity flow to small-area, high-velocity flow, such as flow exiting a structure and entering a downstream pipe. This loss is calculated based on the exit pipe's velocity and a contraction coefficient, as follows:

$$h_c = K_c \frac{V_o^2}{2g}$$

Where:
- $h_c$ = Contraction loss (m, ft)
- $K_c$ = Contraction coefficient (unitless)
- $V_O$ = Exit pipe velocity (m/s, ft/s)
- $g$ = Gravitational acceleration constant (m/s², ft/s²)

The contraction coefficient defaults to the AASHTO documented value of 0.25, but can be changed by the user.

### *AASHTO Bend Loss*

$$h_b = \frac{V_o^2}{2g} - \sum\left[\frac{(1-K_i)Q_i}{Q_o}\frac{V_i^2}{2g}\right]$$

Where: $h_b$ = Bend loss (m, ft)

$V_O$ = Outflow pipe velocity (m/s, ft/s)

$Q_O$ = Outflow pipe velocity (m/s, ft/s)

$V_i$ = Inflow pipe velocity (m/s, ft/s)

$Q_i$ = Inflow pipe flow ($m^3/s$, cfs)

g = Gravitational acceleration constant ($m/s^2$, $ft/s^2$)

$K_i$ = Bend factor

**The previous equation is a generalized version of the equation as it appears in the AASHTO manual.**

The program automatically computes a bend factor based on the angles at which the pipes come together. The program's default bend factors are based on Figure 13-12 of the AASHTO manual, but these values, as with other AASHTO coefficients and corrections, can be changed by the user.

### *AASHTO Bend Loss Original Equation*

The structure bend loss is computed for each incoming pipe using the following equation from the AASHTO manual. Losses are computed for each incoming pipe, and the greatest value is used.

$$h_b = K_i \cdot \frac{V_0^2}{2g}$$

Where: $h_b$ = Bend loss (m, ft)

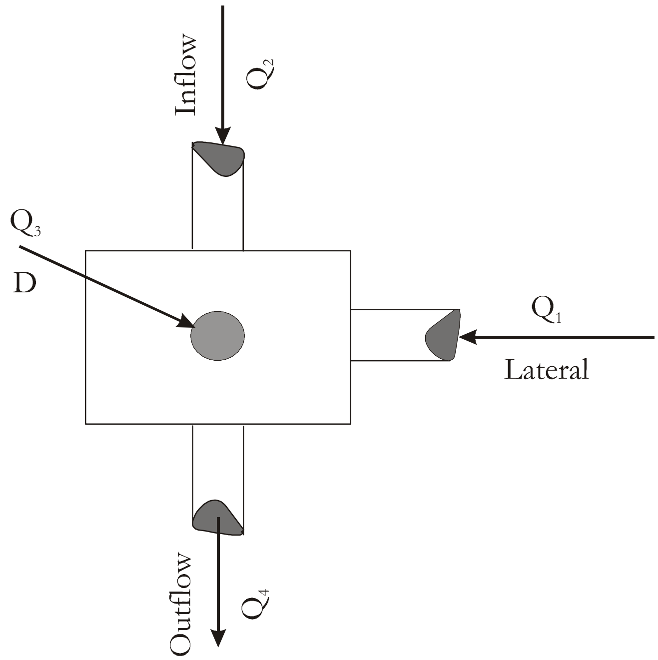
$K_i$ = Bend loss coefficient (unitless)

$V_o$ = Incoming pipe's velocity (m/s, ft/s)

g = Gravitational acceleration constant ($m/s^2$, $ft/s^2$)

The AASHTO manual also documents another bend loss method shown in the following equation. The authors of the AASHTO manual agree that either equation is acceptable. Because of the following equation's tendency to compute negative bend losses in certain cases, we decided to use the above equation exclusively within this program.

$$H_{i_2} = \frac{Q_4 V_4^2 - Q_1 V_1^2 - Q_2 V_2^2 + KQ_q V_1^2}{2gQ_4}$$




### *AASHTO Expansion Loss*

Expansion losses are encountered when small-area, high-velocity flow meets a large-area, low-velocity flow, such as a pipe discharging into a structure. To compute this loss, the following equation is used:

$$h_e = K_e \cdot \frac{V_s^2}{2g}$$

Where:
- $h_e$ = Expansion loss (m, ft)
- $K_e$ = Expansion coefficient (unitless)
- $V_s$ = Most significant incoming pipe's velocity (m/s, ft/s)
- $g$ = Gravitational acceleration constant ($m/s^2$, $ft/s^2$)

The most significant pipe is the pipe that has the greatest product of velocity and discharge, omitting any pipes that have a discharge less than 10% of the structure's outflow. The expansion coefficient defaults to the AASHTO documented value of 0.35, but can be changed by the user.

### *AASHTO Correction For Non-Piped Flow*

If non-piped flow accounts for 10% or more of the total structure outflow, a correction factor is applied to the total loss. By default, this value is a 30% increase in headloss (a factor of 1.3) as documented in the AASHTO manual, but can be changed by the user.

### *AASHTO Correction for Shaping*

If the bottom of the structure is shaped to facilitate smoother transitions from inflow pipes to the discharge pipe, a correction factor can be applied to the total loss. By default, this value is a 50% reduction (a factor of 0.5) as documented in the AASHTO manual, but can be changed by the user.

## B.4.2 Minor Losses

Minor losses in pressure pipes are caused by localized areas of increased turbulence that create a drop in the energy and hydraulic grades at that point in the system. The magnitude of these losses is dependent primarily upon the shape of the fitting, which directly affects the flow lines in the pipe.

The equation most commonly used for determining the loss in a fitting, valve, meter, or other localized component is:

$$h_m = K\frac{V^2}{2g}$$

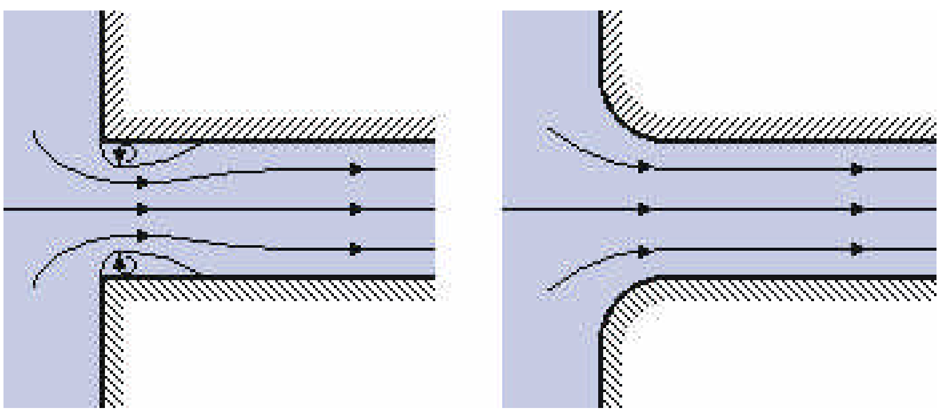
Where:

- $h_m$ = Loss due to the minor loss element (m, ft)
- $V$ = Velocity (m/s, ft/s)
- $g$ = Gravitational acceleration constant ($m/s^2$, $ft/s^2$)
- $K$ = Loss coefficient for the specific fitting

Typical values for the fitting loss coefficient are included in the Fittings Table at the end of this chapter.

Generally speaking, more gradual transitions create smoother flow lines and smaller headlosses. For example, the figure below shows the effects of a radius on typical pipe entrance flow lines.

Flow Lines at Entrance



### Fitting Loss Coefficients

For similar fittings, the K-value is highly dependent on things such as bend radius and contraction ratios.

Typical Fitting K Coefficients

| Fitting | K Value | Fitting | K Value |
|---|---|---|---|
| Pipe Entrance | | 90° Smooth Bend | |
| Bellmouth | 0.03 - 0.05 | Bend radius / D = 4 | 0.16 - 0.18 |
| Rounded | 0.12 - 0.25 | Bend radius / D = 2 | 0.19 - 0.25 |
| Sharp Edged | 0.50 | Bend radius / D = 1 | 0.35 - 0.40 |
| Projecting | 0.80 | | |
| | | Mitered Bend | |
| Contraction - Sudden | | $\theta = 15°$ | 0.05 |
| $D_2/D_1 = 0.80$ | 0.18 | $\theta = 30°$ | 0.10 |
| $D_2/D_1 = 0.50$ | 0.37 | $\theta = 45°$ | 0.20 |
| $D_2/D_1 = 0.20$ | 0.49 | $\theta = 60°$ | 0.35 |
| Contraction - Conical | | $\theta = 90°$ | 0.80 |
| $D_2/D_1 = 0.80$ | 0.05 | | |
| $D_2/D_1 = 0.50$ | 0.07 | Tee | |
| $D_2/D_1 = 0.20$ | 0.08 | Line Flow | 0.30 - 0.40 |
| | | Branch Flow | 0.75 - 1.80 |
| Expansion - Sudden | | | |
| $D_2/D_1 = 0.80$ | 0.16 | Cross | |
| $D_2/D_1 = 0.50$ | 0.57 | Line Flow | 0.50 |
| $D_2/D_1 = 0.20$ | 0.92 | Branch Flow | 0.75 |
| Expansion - Conical | | | |
| $D_2/D_1 = 0.80$ | 0.03 | 45° Wye | |
| $D_2/D_1 = 0.50$ | 0.08 | Line Flow | 0.30 |
| $D_2/D_1 = 0.20$ | 0.13 | Branch Flow | 0.50 |

# B .5 Pumping Stations and Pressure Sewers

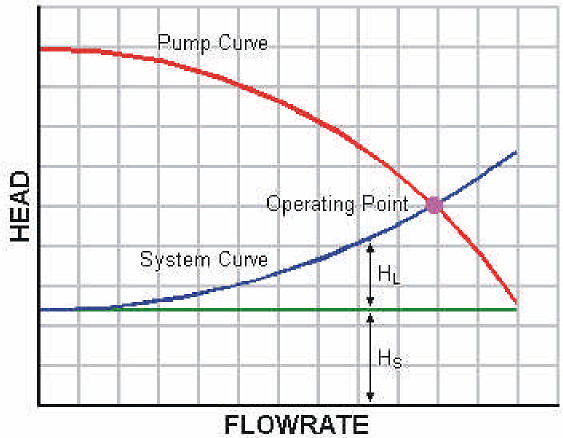
## B.5.1 Pump Theory

Pumps are an integral part of many pressure systems. Pumps add energy, or head gains, to the flow to counteract headlosses and hydraulic grade differentials within the system.

A pump is defined by its characteristic curve, which relates the pump head, or the head added to the system, to the flow rate. This curve is indicative of the ability of the pump to add head at different flow rates. To model behavior of the pump system, additional information is needed to ascertain the actual point at which the pump will be operating.

The system operating point is based on the point at which the pump curve crosses the system curve representing the static lift and headlosses due to friction and minor losses. When these curves are superimposed, the operating point can easily be found. This is shown in the figure below.

System Operating Point




As water surface elevations and demands throughout the system change, the static head (Hs) and headlosses (HL) vary. This changes the location of the system curve, while the pump characteristic curve remains constant. These shifts in the system curve result in a shifting operating point over time.

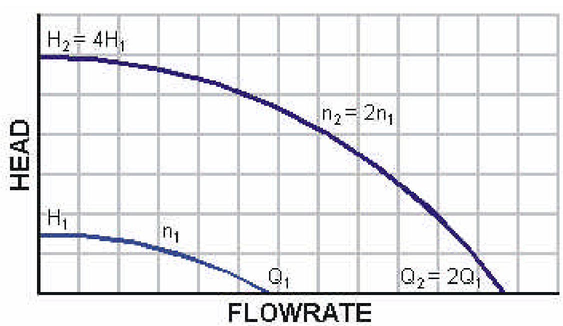
### Variable Speed Pumps

A pump's characteristic curve is fixed for a given motor speed and impeller diameter, but can be determined for any speed and any diameter by applying the affinity laws. For variable speed pumps, these affinity laws are presented as:

$$\frac{Q_1}{Q_2} = \frac{n_1}{n_2} \quad \text{and} \quad \frac{h_1}{h_2} = \left(\frac{n_1}{n_2}\right)^2$$

Where:
- Q = Pump flowrate ($m^3/s$, cfs)
- h = Pump head (m, ft)
- n = Pump speed (rpm)

Effect of Relative Speed on Pump Curve



**Constant Horsepower Pumps**

During preliminary studies, the exact characteristics of the constant horsepower pump may not be known. In these cases, the assumption is often made that the pump is adding energy to the water at a constant rate. Based on power-head-flowrate relationships for pumps, the operating point of the pump can then be determined. Although this assumption is useful for some applications, a constant horsepower pump should only be used for preliminary studies.

## B.5.2 Pump Type

This software currently models six different types of pumps:

- **Design Point (One-Point) -** A pump can be defined by a single design point (Hd @ Qd). From this point, the curve's interception with the head and discharge axes is computed as Ho = 1.33•Hd and Qo = 2.00•Qd. This type of pump is also useful for preliminary designs, but should not be used for final analysis.
- **Standard (Three-Point)** - This pump curve is defined by three points - the shutoff head (pump head at zero discharge), the design point (as with the single-point pump), and the maximum operating point (the highest discharge at which the pump performs predictably).
- **Standard Extended** - The same as the standard three-point pump, but with an extended point at the zero pump head point. This is automatically calculated by the program.
- **Custom Extended** - The custom extended pump is similar to the standard extended pump, but allows you to enter the discharge at zero pump head.
- **Multiple Point -** This option allows you to define a custom rating curve for a pump. The pump curve is defined by entering points for discharge rates at various heads. Since the general pump equation, shown below, is used to simulate the pump during the network computations, the user-defined pump curve points are used to solve for coefficients in the general pump equation:

$$Y = A - (B \times Q^C)$$

$$Y = A - (B \times Q^C)$$

Where: $Y$ = Head (m, ft)
$Q$ = Discharge ($m^3/s$, cfs)
A, B, C = Pump curve coefficients

The Levenberg-Marquardt Method is used to solve for A, B and C based on the given multiple-point rating curve.

- **Constant Power -** These pumps may be useful for preliminary designs and estimating pump size, but should not be used for any analysis for which more accurate results are desired.

**Whenever possible, avoid using constant power or design point pumps. They are often enticing because they require less work on behalf of the engineer, but they are much less accurate than a pump curve based on several representative points.**

**It is not necessary to place a check valve on the pipe immediately downstream of a pump, because pumps have built in check valves that prevent reverse flow.**

## B.5.3 Conservation of Mass and Energy

Conservation of Mass

At any node in a system containing incompressible fluid, the total volumetric or mass flows in must equal the flows out, less the change in storage. Separating these into flows from connecting pipes, demands, and storage, we obtain:

$$\sum Q_{IN}\Delta t = \sum Q_{OUT}\Delta t + \Delta V_S$$

Where: QIN = Total flow into the node ($m^3$/s, cfs)

QOUT = Total demand at the node ($m^3$/s, cfs)

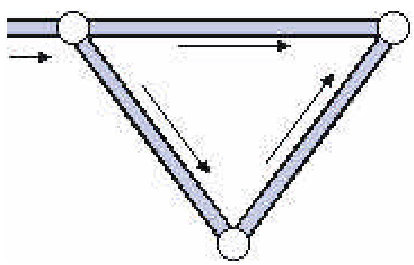
ΔVS = Change in storage volume ($m^3$, $ft^3$)

Δt = Change in time (s)

Conservation of Energy

The conservation of energy principle states that the headlosses through the system must balance at each point. For pressure networks, this means that the total headloss between any two nodes in the system must be the same regardless of what path is taken between the two points. The headloss must be sign consistent with the assumed flow direction (i.e. gain head when proceeding opposite the flow and lose head when proceeding with the flow).

Conservation of Energy



The same basic principle can be applied to any path between two points. As shown in the figure above, the combined headloss around a loop must equal zero in order to achieve the same hydraulic grade as at the beginning.

## B.5.4 The Gradient Algorithm

The gradient algorithm for the solution of pipe networks is formulated upon the full set of system equations that model both heads and flows. Since both continuity and energy are balanced and solved with each iteration, the method is theoretically guaranteed to deliver the same level of accuracy observed and expected in other well known algorithms such as the Simultaneous Path Adjustment Method (Fowler) and the Linear Theory Method (Wood).

In addition, there are a number of other advantages that this method has over other algorithms for the solution of pipe network systems:

- The method can directly solve both looped and partly branched networks. This gives it a computational advantage over some loop-based algorithms, such as Simultaneous Path, which require the reformulation of the network into equivalent looped networks or pseudo-loops.
- Using the method avoids the post-computation step of loop and path definition, which adds significantly to the overhead of system computation.

- The method is not numerically unstable when the system becomes disconnected by check valves, pressure regulating valves, or modeler's error. The loop and path methods fail in these situations.
- The structure of the generated system of equations allows the use of extremely fast and reliable sparse matrix solvers.

The derivation of the Gradient Algorithm starts with two matrices and ends as a working system of equations.

## B.5.5 Derivation of the Gradient Algorithm

Given a network defined by N unknown head nodes, P links of unknown flow, and B boundary or fixed head nodes, the network topology can be expressed in two incidence matrices:

$A_{12} = A_{21}{}^T$ (P x N) Unknown head nodes incidence matrix

and

$A_{10} = A_{01}{}^T$ (P x B) Fixed head nodes incidence matrix

The following convention is used to assign matrix values:

$A_{12}(i, j) =$ 1, 0, or -1 if flow of pipe i enters, is not connected, or leaves node j, respectively.

Assigned nodal demands are given by:

$q^T = [q_1, q_2, ..., q_N]$ (1 x N) nodal demand vector

Assigned boundary nodal heads are given by:

$H_f{}^T = [H_{f1}, H_{f2}, ...H_{fB}]$ (1 x B) fixed nodal head vector

The headloss or gain transform is expressed in the matrix:

$F^T(Q) = [f_1, f_2, ..., f_P]$ (1 x P) non-linear laws expressing headlosses in links

$f_i = f_i(Q_i)$

These matrix elements that define known or iterative network state can be used to compute the final steady-state network represented by the matrix quantities for unknown flow and unknown nodal head.

Unknown link flow quantities are defined by:

$Q^T = [Q_1, Q_2, ..., Q_P]$ (1 x P) unknown link flow rate vector

Unknown nodal heads are defined by:

$$H^T = [H_1, H_2, \ldots, H_N]$$ (1 x N) unknown nodal head vector

These topologic and quantity matrices can be formulated into the generalized matrix expression using the laws of energy and mass conservation:

$$A_{12}H + F(Q) = -A_{10}H_f$$

$$A_{12}Q = q$$

A second diagonal matrix that implements the vectorized head change coefficients is introduced. It is generalized for Hazen-Williams friction losses in this case:

$$A_{11} = \begin{bmatrix} R_1|Q_1|^{n_1-1} & & & & \\ & R_2|Q_2|^{n_2-1} & & & \\ & & \ldots & & \\ & & & \ldots & \\ & & & & R_P|Q_P|^{n_P-1} \end{bmatrix}$$

This yields the full expression of the network response in matrix form:

$$\begin{bmatrix} A_{11} & A_{12} \\ A_{21} & 0 \end{bmatrix} \begin{bmatrix} Q \\ H \end{bmatrix} = \begin{bmatrix} -A_{10}H_f \\ q \end{bmatrix}$$

To solve the system of non-linear equations, the Newton-Raphson iterative scheme can be obtained by differentiating both sides of the equation with respect to Q and H to get:

$$\begin{bmatrix} NA_{11} & A_{12} \\ A_{21} & 0 \end{bmatrix} \begin{bmatrix} dQ \\ dH \end{bmatrix} = \begin{bmatrix} -dE \\ dq \end{bmatrix}$$

with

$$N = \begin{bmatrix} n_1 & & & \\ & n_2 & & \\ & & \ldots & \\ & & & n_P \end{bmatrix}$$

The final recursive form of the Newton-Raphson algorithm can now be derived after matrix inversion and various algebraic manipulations and substitutions (not presented here). The working system of equations for each solution iteration, k, is given by:

$$H^{k+1} = -(A_{21}N^{-1}A_{11}{}^{-1}A_{12})^{-1}\left\{A_{21}N^{-1}(Q^k + A_{11}{}^{-1}A_{10}H_f) + (q - A_{21}Q^k)\right\}$$

$$Q^{k+1} = (1 - N^{-1})Q^k - N^{-1}A_{11}{}^{-1}(A_{12}H^{k+1} + A_{10}H_f)$$

The solution for each unknown nodal head for each time iteration is computationally intensive. This high speed solution utilizes a highly optimized sparse matrix solver that is specifically tailored to the structure of this matrix system of equations.

Sources:

Todini, E. and S. Pilati, "A gradient Algorithm for the Analysis of Pipe Networks", *Computer Applications in Water Supply, Vol. 1 - Systems Analysis and Simulation*, ed. By Bryan Coulbeck and Chun-Hou Orr, Research Studies Press LTD, Letchworth, Hertfordshire, England.

## B.5.6 The Linear System Equation Solver

The Conjugate Gradient method is one method that, in theory, converges to an exact solution in a limited number of steps. The Gradient working equation can be expressed for the pressure network system of equations as:

$$Ax = b$$

where:

$$x = H^{k+1}$$

$$b = -\left\{A_{21}N^{-1}(Q^k + A_{11}{}^{-1}A_{10}H_f) + (q - A_{21}Q^k)\right\}$$

The structure of the system matrix A at the point of solution is:

$$A = A_{21}(NA_{11})^{-1}A_{12} = A_{21}DA_{12}$$

and it can be seen that the nature of the topological matrix components yield a total working matrix A that is:

- Symmetric
- Positive definite
- Stieltjes type

Because of the symmetry, the number of non-zero elements to be retained in the matrix equals the number of nodes plus the number of links. This results in a low density, highly sparse matrix form. It follows that an iterative solution scheme would be preferred over direct matrix inversion, in order to avoid matrix fill-in which serves to increase the computational effort.

Because the system is symmetric and positive definite, a Cholesky factorization can be performed to give:

$$A = LL^T$$

where L is lower triangular with positive diagonal elements. Making the Cholesky factorization allows the system to be solved in two steps:

$$y = L^{-1}b$$

$$x = (L^T)^{-1}y$$

The use of this approach over more general sparse matrix solvers that implement traditional Gaussian elimination methods without consideration to matrix symmetry is preferred, since performance gains are considerable. The algorithm utilized in this software solves the system of equations using a variant of Cholesky's method which has been optimized to reduce fill-in of the factorization matrix, thus minimizing storage and reducing overall computational effort.

# B .6 Extended Period Simulations

## B.6.1 Extended Period Simulations Overview

The Extended Period Simulation (EPS) models how a sewer network will behave over time. This type of analysis allows the user to model wet wells filling and draining, how pumps toggle on and off, and how pressures, hydraulic grades, and flow rates change throughout the system in response to varying loading conditions and in response to automatic control strategies formulated by the modeler. In SewerCAD the algorithm proceeds in a general downstream direction proceeding towards the outfall and occurs in the following steps:

1. The analysis begins in the gravity portion of the network. All hydrographs are generated entering into the gravity system and successively routed and summed as the flows approach the bounding wet well. Ultimately, the total inflow hydrograph to the wet well is determined.
2. Knowing the inflow to the wet well, the pressure calculations for the force main system bounded by the wet well are performed. In addition to flow velocities and pressures, the levels in the wet well are determined over time.
3. SewerCAD then returns to the gravity portion of the network discussed in step 1. The hydraulics and HGL profiles are calculated throughout the gravity system for each time step using the known level of the wet well as the boundary condition for the backwater analysis.

The process then repeats, continuing to the systems downstream of the pressure network until an outlet is reached.

## B.6.2 Routing Overview

As a hydrograph flows through a conduit it undergoes changes in shape and temporal distribution caused by translation and storage effects.

SewerCAD uses two methods to determine the shape and distribution of a hydrograph routed through a gravity pipe.

- Convex Routing
- Weighted Translation Routing

## B.6.3 Convex Routing

The underlying assumption of the convex routing method is that the routed outflow for a time step is based on the inflow and outflow for the previous time step. Each outflow ordinate is calculated as:

$$O_{t+\Delta t} = cI_t + (1-c)O_t$$

| | | | |
|---|---|---|---|
| Where: | $O_{t+\Delta t}$ | = | Outflow at time t + Δt |
| | t | = | Current time (s, min) |
| | Δt | = | Hydrologic time step (s, min) |

| | | |
|---|---|---|
| c | = | Convex routing coefficient |
| $I_t$ | = | Inflow at time t (l/s, gpm) |
| $O_t$ | = | Outflow at time t (l/s, gpm) |

The convex routing coefficient is essentially a ratio of the hydrologic time step and representative flow travel time through the pipe and is calculated as follows:

$$c = \Delta t \frac{V}{L} = \frac{\Delta t}{t_t}$$

| Where: | | | |
|---|---|---|---|
| | $\Delta t$ | = | Hydrologic time step (s) |
| | $t_t$ | = | Travel time (s) |
| | V | = | Velocity established for representative flow. (m/s, ft/s) |
| | L | = | Length of pipe (m, ft) |

The velocity used to calculate the coefficient is either the normal velocity or full flow velocity generated for a user-specified percentage of the peak of the inflow hydrograph. In other words, if the percentage of the peak flow is greater than the capacity of the pipe then the full-flow velocity is used. If the percentage of the peak flow is less than the capacity the flow velocity for normal depth is used.

**You can specify the percentage of the peak flow, which is used to calculate the Convex Routing c coefficient for each pipe, by clicking the Go button and then clicking the Options button. Then click the Convex tab of the Calculation Options dialog. The values typically range between 50% and 75%.**

The higher the percentage of flow the faster the velocity used to calculate the convex routing coefficient, hence the closer the routed hydrograph will be to a pure translation of the inflow hydrograph.

The user-specified percentage can be modified in the calculation options. A typical value is around 75 % but can be modified for oddly shaped hydrographs with sharp uncharacteristic peaks or for calibration purposes.

### B.6.4 Weighted Translation Routing

The Convex Routing method is only valid when the Convex Routing coefficient, c is less than 1 or when the hydrologic time step is less than the calculated travel time. In certain cases where the travel time exceeds the hydrologic time step, SewerCAD automatically uses an alternate method of routing.

Each ordinate of the outflow hydrograph is derived from a weighted average of the ordinates for the current and previous time steps of the inflow hydrograph. The weights are calculated based on the Convex Routing coefficient.

Each ordinate of the outflow hydrograph is calculated as follows:

$$O_t = \frac{1}{c} I_{t-\Delta t} + \left(1 - \frac{1}{c}\right) I_t$$

| Where | | | |
|---|---|---|---|
| | $O_t$ | = | Outflow at current time step (l/s, gpm) |
| | c | = | Convex Routing coefficient |
| | $I_{t-\Delta t}$ | = | Inflow at previous time step (l/s, gpm) |
| | $I_t$ | = | Inflow at current time step (l/s, gpm) |

### B.6.5 Hydrologic and Hydraulic Time Steps

SewerCAD uses two distinct time steps when running an Extended Period Simulation.

- **Hydrologic Time Step** - This time step is used to calculate the routed hydrographs and represents the time increment of all hydrographs generated during the analysis. The hydrologic time step is also used as the calculation increment for the pressure calculations.
- **Hydraulic Time Step** - This time step represents how often the hydraulic calculations are performed. Flows are interpolated off the previously generated hydrographs using the hydraulic time step and are used to perform the gradually varied flow analyses for that time step.

The hydrologic time step should be less than or equal to the hydraulic time step. The hydraulic time step should be a multiple of the hydrologic time step.

**The Hydrologic and Hydraulic Time Steps can be modified in the Extended Period Simulation section of the Scenario / Go dialog.**

## B .7 Transitioning Between Gravity and Pressure Networks

### B.7.1 Overview

This section describes the major distinctions between gravity pipes and force mains in SewerCAD. It also describes how flow and the hydraulic grade transition between force mains and gravity pipes and vice versa.

### B.7.2 Identifying Gravity Pipes and Force Mains

Superficially, SewerCAD depicts a force main (pressure pipe) as a single line, and a gravity pipe as two parallel lines.

The difference can also be recognized by the pipe's context in the network. The pipes in a gravity system must all converge on a single termination point in a classic tree structure. Multiple pipes can enter into a single gravity node, but only one may exit. The gravity subnetwork can either terminate on a wet well, or an outlet.

The force mains can be much more complex with loops and multiple outlet points. The pressure subnetwork can terminate on an outlet, a manhole, or a junction chamber.

### B.7.3 Direction of Flow in Gravity and Pressure Systems

In gravity pipes flow will always travel towards the termination point of the gravity network. Once the gravity pipes are drawn in the network and connected to an outlet or wet well, all the pipes will orient themselves such that downstream points toward the termination point.

Pressure systems are usually designed such that flow will travel from the wet well to an outfall point at either an outlet or a gravity system. If the elevation is too high at that point, SewerCAD will allow flow to travel backwards from the gravity system to the pressure network. There are no programmatic limitations as to the direction of flow in the force main system.

## B.7.4 Transitioning From Gravity Pipes to Force Mains

### Overview

The only way to transition between a gravity pipe and a force main in SewerCAD is through an intermediate wet well. This establishes a boundary condition for both the connecting systems. During a Steady State analysis the wet well level can be calculated based on generating the required HGL for the outflow to either match or exceed the inflow, or the wet well level can be fixed to a user-specified level.

During an Extended Period Simulation the wet well level is determined for a time step by calculating the change in storage over time.

The inflow into the wet well is determined by summing all loads flowing to that wet well.

### Hydraulic (HGL) Transition from Gravity to Pressure Network

Gravity hydraulic calculations upstream of a wet well are based on the wet well hydraulic grade, just as they are for standard calculations within gravity systems.

During a Steady State analysis there is a difference, however, in determining the hydraulic grade within the wet well itself. The wet well level may be set by the user to either be fixed or not fixed.

#### *Fixed Wet Well Level during Steady State Analysis*

If the wet well level is fixed, the wet well’s starting hydraulic grade is used for pressure calculations. No adjustments are made, and this grade is used as the tailwater grade for the upstream gravity systems.

#### *Non-Fixed Wet Well Level during Steady State Analysis*

If the wet well level is not fixed, the pressure calculations will attempt to balance the wet well level such that the total flow out of the wet well is equal to or greater than the total flow into the wet well.

The wet well’s starting grade is used for the first iteration. If the calculated flows out of each wet well are greater than or equal to each wet well’s incoming flow, the iterations stop there. If not, the wet well levels are increased by the increment specified in the calculation options, and the pressure subnetwork is recalculated. When the wet well level is increased, it changes the static heads and increases the discharge for connected pumps, and may also trigger additional pumps to turn on.

This process continues until the level in each non-fixed wet well either meets the flow criteria, or is prevented by rising to the maximum elevation of the wet well.

#### *Wet Well Level During Extended Period Simulations*

During an Extended Period Simulation the fixed wet well level options are not available. The wet well level for a time step is actually determined by the change in storage due to inflows and outflows over a single time step.

### Hydrologic (Flow) Transition from Gravity to Pressure Network

The hydrologic transition from a gravity system into a pressure system is quite simple. Gravity loads upstream from the wet well are accumulated and combined with the wet well’s local load to determine the total load entering the pressure system at that location. This can be imagined as the total load "dumping into" the wet well.

## B.7.5 Transitioning From Force Mains to Gravity Elements

### Overview

Force mains can empty directly into the gravity system via a manhole, or a junction chamber, or they can terminate at an outlet. During a Steady State analysis the flow entering the gravity system can either be only the specified sanitary and wet weather loads, or the load generated based on the pumps, the wet well level and the hydraulic characteristics of the force main system.

During an Extended Period Simulation flows to the gravity system are solely determined based on the hydraulic characteristics of the force main system.

If the force main empties into an outlet the HGL boundary is determined by the elevation of the user specified-tailwater. The HGL boundary between the force main and gravity system is determined by the elevation of the force main at the boundary point.

## Hydraulic (HGL) Transition from Pressure to Gravity Network

If the force main empties into and outlet element, then the hydraulic grade will be the higher of the set tailwater elevation or the crown of the pressure pipe.

The hydraulic grade at a manhole or a junction chamber downstream from a discharging force main is not considered when performing pressure calculations. Instead, the boundary hydraulic grade is assumed to be equal to the crown elevation of the discharge pipe.

For example, if a 150 mm pipe discharges at an invert elevation of 10.000 m, the hydraulic grade that is used as the boundary condition for the pressure system is 10.150 m.

This assumption is absolutely valid and conservative when the hydraulic grade line in the gravity element is below the crown of the pipe.

This is generally a valid assumption when the hydraulic grade line is above the crown of the pipe as the head at the entrance of the gravity network is often insignificant compared to the friction losses incurred through the force mains.

## Hydrologic (Flow) Transition from Pressure to Gravity Network

The hydrologic transition from a pressure system into a gravity system during a Steady State analysis is somewhat complicated, with different desired behaviors for different analysis purposes. If the model is built for direct analysis of flows in the system, it is most likely that instantaneous pumped flows are important, and the downstream gravity loads should include the pumped discharge from the pressure system. If the analysis is for large-scale planning purposes, however, it is more likely that instantaneous pumping rates are not as important. In these cases, the downstream gravity system should be analyzed based on the total contributing population, area, or other factors for determining peak flows.

### *Hydrologic Transition from Pressure to Gravity During Steady State Analysis*

#### *Conserving Pumped Flow*

When instantaneous pumped flows are a concern in the downstream gravity system, the calculation option to conserve pumped flow should be used. This is found on the **Pressure Hydraulics Options** tab of the **Calculation Options** dialog. When this option is used, the contributing load components from the pressure system are ignored, and a load of type "Pumped" is added to the gravity system with the magnitude of the flow in the discharging pressure pipe. If the flow in the pressure pipe is such that the gravity system would be draining into the pressure system, no load is transferred to that gravity node.

#### *Conserving Load Components*

When instantaneous pumped flows are not a concern in the downstream gravity system, the calculation option to conserve load components (not pumped flow) should be used. When conserving load components, the total contributing components to the pressure system (population, area, and so forth) are considered. The total contributing load is distributed to the downstream gravity systems proportionally to the pressure pipe discharge rates. If the flow in a pressure pipe is such that the gravity system would be draining into the pressure system, no load is transferred to that gravity node.

### *Comparison of Pressure Load Routing*

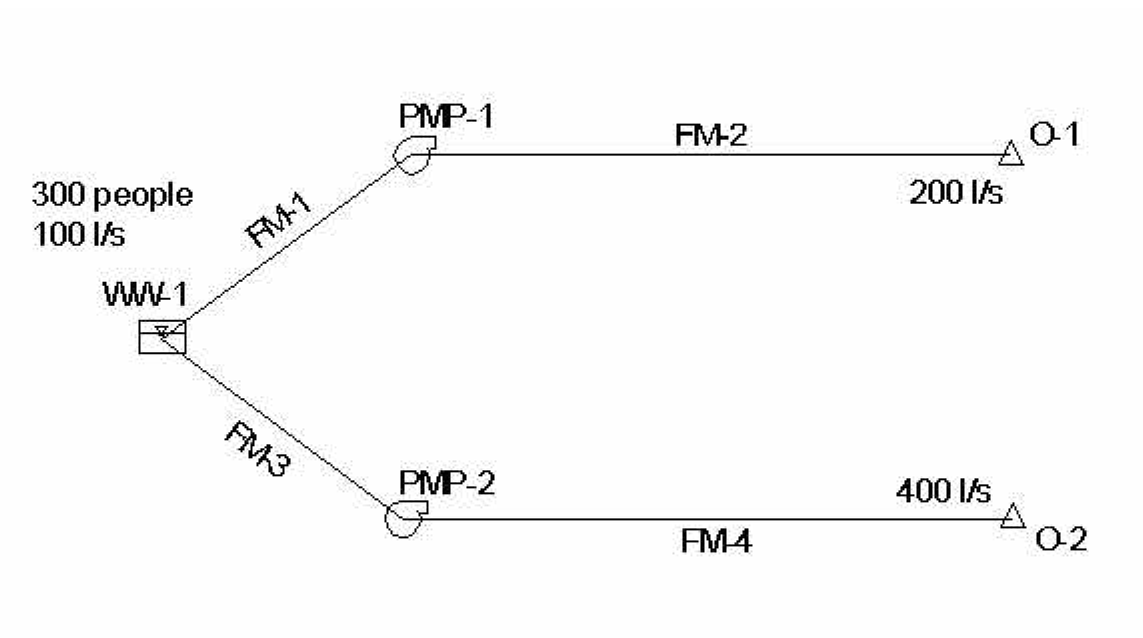
Consider the system below with the following load equivalents:

300 people = 100 l/s

200 people = 70 l/s

100 people = 50 l/s

Network System



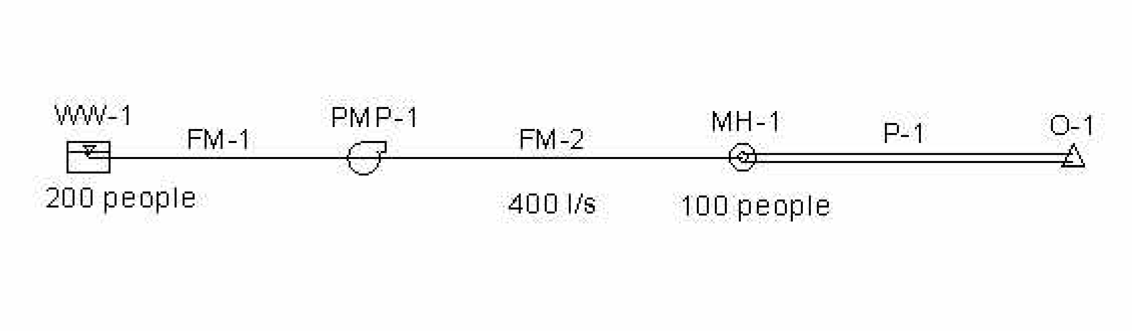

The total load population contributing to the load entering the pressure system is 300 people, equivalent to a flow of 100 liters per second. The operating points of each pump are such that the discharges into the downstream gravity systems are 200 liters per second and 400 liters per second. This is more than the rate of flow entering the wet well.

When conserving pumped flow, the flows in each discharging force main are transferred directly to the downstream gravity system, resulting in a pumped load of 200 l/s at O-1 and a pumped load of 400 l/s at O-2.

When conserving load components, the 300 people contributing to the load at the wet well are split proportionally to the downstream gravity systems (1/3 and 2/3). This results in a load of 100 people (50 l/s) at O-1, and 200 people (70 l/s) at O-2.

The differences can also be seen for the following system, with the same load equivalents as above.

Network System




When conserving pumped loads, the downstream gravity load is computed as:

400 l/s + 100 people = 400 l/s + 50 l/s = 450 l/s

When conserving load components, the downstream gravity load is computed as:

200 people + 100 people = 300 people = 100 l/s

Note that for this piping configuration (one source of loading for the pressure system, one possible discharge point for the pressure system), the downstream load will always be 100 l/s (300 people) regardless of the actual pump operating point.

#### *Hydrologic Transition from Pressure to Gravity During Extended Period Simulations*

Since the wet well level will vary over time, based on the inflow and outflow, no distinction is made between standard loads and pumped loads during an Extended Period Simulation. The loads will be transferred through the force main system based on the hydraulic characteristics of the pumps and the system headlosses.

If the elevation of the force main emptying into the gravity system is high enough that flow goes backward from the gravity element during an Extended Period Simulation, the negative portion of the hydrograph going through the gravity system at that point will be truncated. Only positive flows are analyzed in the gravity system. Negative flows are assumed to be 0. (See the Hydraulic Transition from Pressure to Gravity Network for more information)

## B .8 Constraint Based Automatic Design

### B.8.1 Gravity Pipes and Structures Design

This program allows you to automatically design gravity piping and structures. The design is flexible enough to allow you to specify the elements to be designed, from a single pipe size to the entire system, or anything in between.

The design algorithm adjusts invert elevations and the section size of the pipe to meet several constraints, such as allowable ranges of slope, velocity and cover. In general, the design algorithm attempts to minimize pipe size and excavation, which is typically the most expensive part of installing sewer piping and structures.

Some of the other things that are considered include:

- Pipe Matching
- Offset Matching
- Drop Structures

- Structure Sump Elevations

The designed pipe will be the smallest available section size from the **Engineering Library** that meets the constraints and has a capacity greater than its discharge. In a situation where there are no pipe sizes with adequate capacity, the largest available size will be used.

## B.8.2 Part Full Design

Pipes are designed such that the capacity is greater than the calculated discharge. For standard designs, this capacity is based on full pipe, normal depth - that is, the flow in the pipe when the depth is 100% of the pipe rise.

With partially full design, the designed capacity of the pipe is for a design depth that is only a portion of the pipe rise. In other words, a pipe that is designed for 50% full will be selected based on a depth of half of the pipe's rise.

For example, consider a circular pipe with the following characteristics:

Slope = 0.01 m/m

Roughness n = 0.013

Required flow = 100 l/s

The following table presents several typical section sizes, with their capacities at various depths.

Flow Capacities

| **Circular Section Size** | **100% Full** | | **80% Full** | | **50% Full** | |
|---|---|---|---|---|---|---|
| **Nominal Diameter** | **Depth (mm)** | **Capacity (l/s)** | **Depth (mm)** | **Capacity (l/s)** | **Depth (mm)** | **Capacity (l/s)** |
| **300 mm** | **300** | **101** | **240** | **99** | **150** | **50** |
| **375 mm** | **375** | **183** | **300** | **179** | **188** | **91** |
| **450 mm** | **450** | **297** | **360** | **291** | **225** | **149** |

Depending on the selected percent-full, the smallest available pipe could be for any of the bold values above. Obviously, if the design percentage were something different, an even larger section may be required.

Hydraulically, the capacity at a percentage of pipe rise is generally not equal to that percentage of the full pipe capacity. As can be seen in the table above, 80%-full capacity does not equal 80% of the 100%-full capacity.

For sections that are vertically symmetrical, 50% full is a special case where the wetted perimeter and area are both half that of full flow. This means that the hydraulic radius and velocity are the same for half-full and full flow, resulting in a highly special condition where the 50%-full capacity is actually equal to one half of the 100%-full capacity.

## B.8.3 Allow Multiple Sections

Situations may be encountered where the desired capacity cannot be met with a single pipe, due to a limiting maximum section rise, a lack of larger available pipes, or other restrictions. For these situations, the pipe can be designed with multiple barrels. All barrels will have the same physical characteristics.

Multiple barrels will only be used if the design cannot be met by a single available section size, and the pipe allows multiple sections for design. In these cases, the design will increase the number of barrels and attempt to find a section size that meets the capacity, continuing until the capacity is met or the maximum number of barrels is reached.

For example, consider a circular pipe with the following characteristics:

Slope = 0.01 m/m

Roughness n = 0.013

Required flow = 750 l/s

Maximum Design Section Rise = 700 mm

Assume that the design is for 100% full capacity, allowing up to three barrels of the following section sizes:

Design with Multiple Sections

| **Circular Section Size** | **1 Barrel** | | **2 Barrels** | | **3 Barrels** | |
|---|---|---|---|---|---|---|
| **Nominal Diameter** | **Capacity (l/s)** | **Meets Flow?** | **Capacity (l/s)** | **Meets Flow?** | **Capacity (l/s)** | **Meets Flow?** |
| **300 mm** | **101** | **No** | **202** | **No** | **203** | **No** |
| **375 mm** | **183** | **No** | **366** | **No** | **549** | **No** |
| **450 mm** | **297** | **No** | **595** | **No** | **892** | **Yes** |
| **525 mm** | **449** | **No** | **897** | **Yes** | **1346** | **Yes** |
| **600 mm** | **641** | **No** | **1281** | **Yes** | **1922** | **Yes** |

For these conditions, the selected design would use two 525 mm barrels - the smallest section size within the least number of barrels to meet the capacity criteria.

### B.8.4 Limit Section Size

There may be situations in design where it is desired to limit the size of the designed pipe. This may be done to avoid conflicts with obstructions or other utilities, for example. For these situations, The program enables you to limit the maximum section rise that will be selected. A smaller size will be used if possible.

If none of the available design sections have a small enough rise, the smallest one will be used.

### B.8.5 Pipe Matching

When pipes meet at a structure, it is often desirable to have the pipes at approximately the same elevation. To do this, the program allows you to design your pipes to match inverts or crowns. This means that when the design is done (if a valid design was found), all of the designed pipes entering a structure will have the same invert elevation or crown elevation.

### B.8.6 Offset Matching

If an offset value is specified, it represents the desired drop across the structure. The design incorporates this offset, resulting in upstream pipes that are higher than the downstream pipe by the specified offset. Note that all designed upstream pipes will have the same invert or crown elevation.

For example, an offset of 0.1 meter could result in a downstream pipe with an invert of 100.0 meters, and several upstream pipes with invert elevations at 100.1 meters.

## B.8.7 Drop Structures

Drop structures are structures at which the incoming pipes are not all at the same elevation, nor do any of them necessarily match the downstream pipe. Including these structures may help to reduce excavation, since the entire upstream system does not need to be as deep.

The program will only use drop structures if you have chosen to allow them, and if a pipe's maximum slope constraint cannot be met. Otherwise, the upstream system will be designed as needed to maintain the desired slope and velocity constraints, which may require significantly lower pipe elevations.

## B.8.8 Structure Sump Elevations

The program can adjust structure sump elevations to account for the invert elevations of newly designed pipes, and any desired additional sump depth.

For example, if a structure is to be adjusted with a sump depth of 0.5 meters and the lowest pipe invert is 100.0 meters, the structure sump elevation would be set to 99.5 meters.

## B.8.9 Design Priorities

Unfortunately, it is not always possible to automate a design that meets all desired constraints. With this in mind, there are certain priorities that are considered when the automated design is performed. These priorities are in place to try to minimize the effect on existing portions of the system while providing appropriate capacity in the designed pipes.

While this sequence does not go into complete detail regarding the design process, it does indicate the general priorities for the automated design. The priorities, of course, only deal with elements that are being designed. If a pipe has fixed inverts or is not to be designed at all, some or all of these criteria obviously do not apply.

### *A Designed Pipe Should Fit within Adjacent Existing Structures*

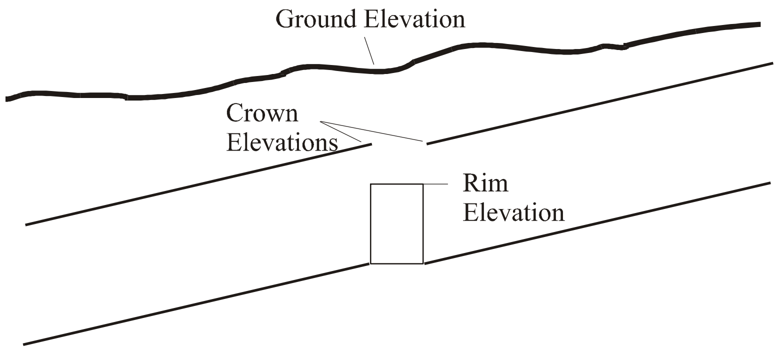
If a pipe connects to an existing structure, the pipe rise should be completely within the existing structure. The only time this may be violated is if there are no available section sizes that would not violate that condition (i.e., the existing structure height is so small that all available pipes have rises too big). In this very unlikely condition, the smallest available section size will be selected, with the invert elevation placed at the bottom of the structure.

### *A Designed Pipe Should Not Have a Crown Above an Adjacent Designed Structure*

Where pipe inverts are fixed, it is possible that the required section size would cause the pipe crown to be higher than the top elevation of an adjacent designed structure. If all available pipe section rises are greater than the depth of the pipe invert, the smallest pipe size will be chosen.

> **This situation will only be encountered in situations where the structure's top elevation is set equal to the ground elevation - otherwise, the structure will be designed with a higher top elevation.**

Crown above Adjacent Structure




### *Pipe Capacity Should Be Greater Than the Discharge*

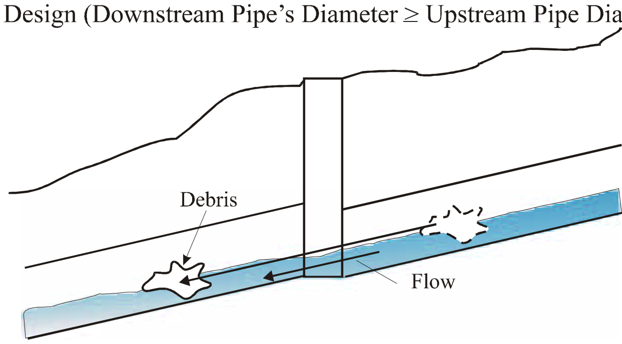
If the pipe is not limited by adjacent structures, the pipe should be sized such that the design capacity is greater than the calculated discharge in the pipe. The design capacity may be based on one or more pipes, flowing full or part-full, depending on user-set design options. If site restrictions or available section limitations result in a situation where no sections meet the required capacity, the largest available size and number of barrels will be chosen.

### *Downstream Pipes Should Be at Least as Large as Upstream Pipes*

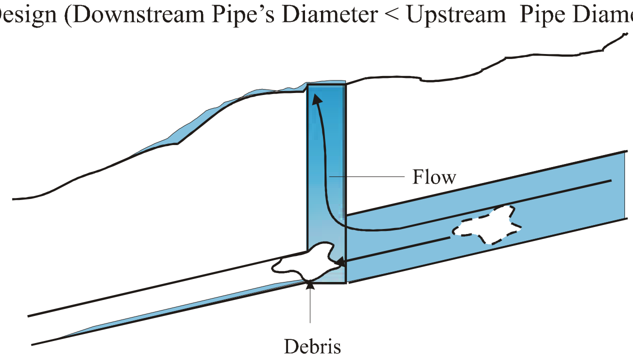
Designs typically avoid sizing downstream pipes smaller than upstream pipes, regardless of differing slope and velocity requirements. One of the primary reasons for this is debris that passes through the upstream pipe could become caught in the connecting structure, clogging the sewer.

Sizing of Pipes

Good Design (Downstream Pipe’s Diameter ≥ Upstream Pipe Diameter)




Bad Design (Downstream Pipe’s Diameter < Upstream Pipe Diameter)



#### *Pipe Matching Criteria Downstream Should Be Met*

Whenever possible, the designed pipe should have its downstream invert set such that the pipe meets the matching criteria, such as matching inverts or crowns. Note that because of higher design priorities, such as the pipe fitting within existing structures, the matching criteria may not always be met.

#### *Minimum Cover Constraint Should Be Met*

Pipe inverts should be set such that the upstream and downstream crowns of the pipe are below the ground elevation by at least the amount of the minimum cover. Note that higher design priorities, such as existing structure locations and matching criteria, may prevent the minimum cover constraint from being met.

#### *Pipe Matching Criteria Upstream Should Be Met*

The upstream invert of the designed pipe should be set to meet the matching criteria of the upstream structure. Higher design priorities, such as minimum cover constraints, may result in a pipe that does not match upstream as desired.

#### *Maximum Slope Constraint Should Be Met*

Wherever possible, the designed pipe should not exceed the desired maximum slope. In some situations, elevation differences across the system may result in a case where a drop structure can be used to offset pipes. This is used instead of a pipe that is too steep, or instead of upstream piping that would require much more excavation. Note that the maximum slope constraint may be violated if higher priority design considerations, such as existing structure location or pipe matching criteria, governs.

#### *Other Constraints and Considerations*

There are many degrees of freedom when designing a piping system. Several constraints that are not mentioned above, such as minimum velocity constraints and minimum slope constraints, may also result in adjustments to the designed pipe. Other constraints may be too limiting, such as maximum cover constraint and maximum velocity, resulting in designed pipes that could violate too many other constraints.

This wide range of choices and priorities emphasizes the need for careful review of any automated design by a professional. It is not always possible to meet every desired condition, so it is very much the responsibility of the engineer to make final judgments and decisions regarding the best design for the client.

### B.8.10 Automatic Design with Hydrograph and Pattern Loads

Automatic designs are run only during a Steady State analysis, which examines only a single instant in time. There are some key behaviors attributed to time-based loads that you may wish to take into account when designing the system.

As described in the *Common Load Types* section of the help, hydrographs can be applied as stead state loads in four different ways (Peak, Average, Minimum, and Zero). If you have hydrograph loading applied to the gravity model, the selected steady state loading option could have a dramatic effect on the ultimate design. For example, when running a design, you may get larger pipes when the inflow loads are based on the peak flows of the hydrographs vs. if they are based on the minimum flows of the hydrographs.

**To access the Steady State Loading options, click the Go button. Then click the Options button. Click the Steady State Loading tab in the Calculation Options dialog.**

Pattern Loads generally consist of an average base load and a diurnal pattern. During a Steady State Analysis, and hence the Design, the pattern is disregarded and the base load is used as the load.

### B.8.11 Constraint Based Warning Messages

The calculated properties of the pipe such as flow velocity, slope, and cover are always being checked against the design constraints regardless of the type of simulation being run.

During a design, you will get warning messages associated with a particular pipe if the algorithm could not attain a solution where all the constraints are met.

If you are running a regular Steady State analysis or an Extended Period Simulation, however, and the constraints are violated (i.e. a velocity in a pipe is higher than the maximum velocity constraint), then, like in **Design** mode, a warning message will also be generated stating a violation as occurred.

These types of warning messages are generated based entirely on the on the user-specified design constraints, and have no affect on the results or the analysis. These constraints are setup as either default design constraints through the **Analysis** menu, or as local constraints through the element editor or the **Design Alternative**. The local design constraints have precedence over the default design constraints.

## B .9 Special Considerations

There are a few special considerations that should be realized when analyzing a sewer system. These are conditions where special assumptions need to be made, or where calculations may seem counter-intuitive at first glance. These considerations include:

- Energy Discontinuity
- Structure Energy Grade
- Design Considerations

### B.9.1 Energy Discontinuity

The program by default uses hydraulic grade as the basis for its hydraulic computations. Energy grade at any given point is then computed by adding the velocity head to the hydraulic grade. Because of this standard practice, energy discontinuities may occasionally occur, such as when pipe size decreases in the downstream direction, or pipe slope increases.

If you wish the calculations to be based on the energy grade line you can modify the *Haestad.ini* file located in the base *Haestad* directory. Find [SWRC] or [STMC] in the file depending on which product you are using, and create a new line below the heading. In the new line enter in the following text typed exactly as printed below.

StructureLossMode=EGL

Save and close the *Haestad.ini* file and reopen the program. To revert back to an HGL based analysis; simply remove the line from the *Haestad.ini* file and save.

Flow discontinuities can also be responsible for energy discontinuities. Since a structure is analyzed based on a different system time than a pipe, a direct comparison of energy grades is not reasonable.

### B.9.2 Structure Energy Grade

The energy grade line (EGL) at the upstream side of a structure is computed based on the characteristics of the structure and its upstream pipes. The reported EGL is generally reported as the lowest EGL of all non-plunging upstream pipes, based on normalized flow values. If there are no non-plunging pipes upstream, the structure's upstream EGL is taken as the higher of the structure's downstream EGL and upstream hydraulic grade line (HGL).

In situations where the structure's upstream EGL is lower than its downstream EGL or upstream HGL, the highest value governs. This rare condition may indicate that the presumed headloss in the structure is not significant enough to produce the expected energy loss. The modeler may accept this as a minor limitation of the hydraulic theory, or may choose to use different structure headloss methods or values.

The reported upstream velocity and velocity head for the structure are based on the difference between the structure's upstream EGL and HGL.

### B.9.3 Design Considerations

As with any automated design, the program's design is intended only as a preliminary step. It will select pipe sizes, inlet lengths (StormCAD only), and pipe invert elevations based on the input provided, but no computer program can match the skills that an experienced engineer has. The modeler should always review any automated design, and should make any changes required to adjust, improve, and otherwise polish the system.

### B.9.4 Reporting Flow Attributes

SewerCAD has many attributes available in the FlexTables, Annotations, Color Coding, and Database/GIS connections on different aspects of flow in a sanitary sewer system. During a Steady State analysis the flow is broken down into categories, (I.e. wet weather, sanitary, known flow and so on). This is done so the program can apply the correct extreme flow methods. During an Extended Period Simulation essentially all the different flows are lumped together into a single hydrograph and the initial categories are disregarded.

In the gravity portion of the system there are two variables that can be used in both Extended Period Simulations and Steady State analyses. These are valuable to know if you are switching back and forth between the two analysis modes, and would like to maintain the same reports, color coding, etc…

- **Total Flow** - Is available for gravity pipes and gravity nodes. In gravity nodes it represents the sum of all the flow exiting the node. In gravity pipes during a Steady State analysis it represents the sum of all the flow entering into the pipe. In an Extended Period Simulation it represents the flow used during a time step's hydraulic analysis and represents a flow point on the pre-routed hydrograph.
- **Diverted Flow Out** - Is available for gravity nodes. Represents the flow exiting the node via a diversion.

In the pressure portion of the system, the **Pressure Flow** attribute for the pressure pipes shows the total amount of flow going through the pipe for a time step, and is available for both Extended Period Simulations and Steady State analyses.

## B .10 Engineer's Reference

**Coefficients**

- Headloss Coefficients for Manholes and Junctions

**Roughness Values**

- Roughness Values, Manning's Equation
- Roughness Values, Kutter's Equation
- Roughness Values, Darcy-Weisbach (Colebrook-White) Equation
- Roughness Values, Hazen-Williams Equation

### B.10.1 Default Kinematic Viscosity

The Kinematic Viscosity is used in determining the friction coefficient in the Darcy-Weisbach Friction Method. The default units are initially set by Haestad Methods.

To override the default value used by Haestad Methods, follow these steps:

1. SAVE your project and Close your application.
2. Using the **Notepad** Accessory in Windows, Open the HAESTAD.INI file located in the HAESTAD directory created when you installed the product.
3. Edit the following line in the section of the HAESTAD.INI file.
4. DefaultKinematicViscosityInMetersSquaredPerSecond=
5. The entry must look **exactly** as shown above with no spaces between attached words and the indicated letters of each word capitalized.
6. The value you wish to use to override the default kinematic viscosity should be inserted after the = sign and must be in meters squared per second.
7. **Save** the HAESTAD.INI file and exit Notepad.
8. Open the program and proceed with your design.

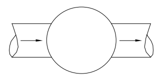
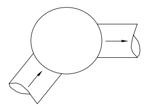
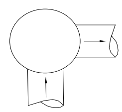
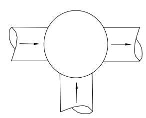
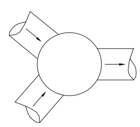
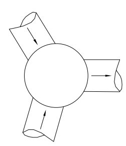
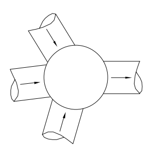
**Changing the default kinematic viscosity value in the HAESTAD.INI file will not change any values for pipes in existing drawings. This value should only be changed before beginning new drawings. Do not use a new value when editing an existing project created with a different kinematic viscosity.**

The program will reset the default value if this entry is left blank in the HAESTAD.INI file.

## B.10.2 Headloss Coefficients for Junctions

These are typical headloss coefficients used in the standard method for estimating headloss through manholes and junctions.

Typical Headloss Coefficients

| Type of Manhole | Diagram | Headloss Coefficient |
|---|---|---|
| Trunkline only with no bend at the junction |  | 0.5 |
| Trunkline only with 45 degree bend at junction |  | 0.6 |
| Trunkline only with 90 degree bend at junction |  | 0.8 |
| Trunkline with one lateral |  | Small 0.6<br>Large 0.7 |
| Two roughly equivalent entrance lines with angle < 90 degrees between lines |  | 0.8 |
| Two roughly equivalent entrance lines with angle > 90 degrees between lines |  | 0.9 |
| Three or more entrance lines |  | 1.0 |

### B.10.3 Roughness Values - Manning's Equation

Commonly used roughness values for different materials are:

Manning’s Coefficients n for Closed Metal Conduits Flowing Partly Full

| Channel Type and Description | Minimum | Normal | Maximum |
|---|---|---|---|
| a. Brass, smooth | 0.009 | 0.010 | 0.013 |
| b. Steel | | | |
| 1. Lockbar and welded | 0.010 | 0.012 | 0.014 |
| 2. Riveted and spiral | 0.013 | 0.016 | 0.017 |
| c. Cast iron | | | |
| 1. Coated | 0.010 | 0.013 | 0.014 |
| 2. Uncoated | 0.011 | 0.014 | 0.016 |
| d. Wrought iron | | | |
| 1. Black | 0.012 | 0.014 | 0.015 |
| 2. Galvanized | 0.013 | 0.016 | 0.017 |
| e. Corrugated metal | | | |
| 1. Subdrain | 0.017 | 0.019 | 0.021 |
| 2. Storm drain | 0.021 | 0.024 | 0.030 |

### B.10.4 Roughness Values - Kutter’s Equation

The roughness values for closed metal conduits flowing partly full for Kutter's Equation are the same as for Manning's Equation.

Some commonly used roughness values for non-metal materials are:

Kutter’s Coefficients n for Closed Non-Metal Conduits Flowing Partly Full

| Channel Type and Description | Minimum | Normal | Maximum |
|---|---|---|---|
| a. Lucite | 0.008 | 0.009 | 0.010 |
| b. Glass | 0.009 | 0.010 | 0.013 |
| c. Cement | | | |
| 1. Neat, surface | 0.010 | 0.011 | 0.013 |
| 2. Mortar | 0.011 | 0.013 | 0.015 |
| d. Concrete | | | |
| 1. Culvert, straight and free of debris | 0.010 | 0.011 | 0.013 |
| 2. Culvert with bends, connections, and some debris | 0.011 | 0.013 | 0.014 |

| Channel Type and Description | Minimum | Normal | Maximum |
|---|---|---|---|
| 3. Finished | 0.011 | 0.012 | 0.014 |
| 4. Sewer with manholes, inlet, etc., straight | 0.013 | 0.015 | 0.017 |
| 5. Unfinished, steel form | 0.012 | 0.013 | 0.014 |
| 6. Unfinished, smooth wood form | 0.012 | 0.014 | 0.016 |
| 7. Unfinished, rough wood form | 0.015 | 0.017 | 0.020 |
| e. Clay | | | |
| 1. Common drainage tile | 0.011 | 0.013 | 0.017 |
| 2. Vitrified sewer | 0.011 | 0.014 | 0.017 |
| 3. Vitrified sewer with manholes, inlet, etc. | 0.013 | 0.015 | 0.017 |
| 4. Vitrified subdrain with open joint | 0.014 | 0.016 | 0.018 |
| f. Brickwork | | | |
| 1. Glazed | 0.011 | 0.013 | 0.015 |
| 2. Lined with cement mortar | 0.012 | 0.015 | 0.017 |
| g. Sanitary sewers coated with sewage slimes, with bends and connections | 0.012 | 0.013 | 0.016 |
| h. Paved invert, sewer, smooth bottom | 0.016 | 0.019 | 0.020 |
| i. Rubble masonry, cemented | 0.018 | 0.025 | 0.030 |

## B.10.5 Roughness Values - Darcy-Weisbach Equation (Colebrook-White)

Commonly used roughness values for different materials are:

Darcy-Weisbach Roughness Heights k for Closed Conduits

| Pipe Material | k (mm) | k (ft) |
|---|---|---|
| Glass, drawn brass, copper (new) | 0.0015 | 0.000005 |
| Seamless commercial steel (new) | 0.004 | 0.000013 |
| Commercial steel (enamel coated) | 0.0048 | 0.000016 |
| Commercial steel (new) | 0.045 | 0.00015 |
| Wrought iron (new) | 0.045 | 0.00015 |
| Asphalted cast iron (new) | 0.12 | 0.0004 |
| Galvanized iron | 0.15 | 0.0005 |
| Cast iron (new) | 0.26 | 0.00085 |
| Concrete (steel forms, smooth) | 0.18 | 0.0006 |
| Concrete (good joints, average) | 0.36 | 0.0012 |
| Concrete (rough, visible, form marks) | 0.60 | 0.002 |
| Riveted steel (new) | 0.9 ~ 9.0 | 0.003 - 0.03 |
| Corrugated metal | 45 | 0.15 |

### B.10.6 Roughness Values - Hazen-Williams Formula

Commonly used roughness values for different materials are:

Hazen-Williams Roughness Coefficients C

| Pipe Material | C |
|---|---|
| Asbestos Cement | 140 |
| Brass | 130-140 |
| Brick sewer | 100 |
| Cast-iron | |
| New, unlined | 130 |
| 10 yr. Old | 107-113 |
| 20 yr. Old | 89-100 |
| 30 yr. Old | 75-90 |
| 40 yr. Old | 64-83 |
| Concrete or concrete lined | |
| Steel forms | 140 |
| Wooden forms | 120 |
| Centrifugally spun | 135 |
| Copper | 130-140 |
| Galvanized iron | 120 |
| Glass | 140 |
| Lead | 130-140 |
| Plastic | 140-150 |
| Steel | |
| Coal-tar enamel, lined | 145-150 |
| New unlined | 140-150 |
| Riveted | 110 |
| Tin | 130 |
| Vitrified clay (good condition) | 110-140 |
| Wood stave (average condition) | 120 |

## B.10.7 Typical Roughness Values for Pressure Pipes

Typical pipe roughness values are shown below. These values may vary depending on the manufacturer, workmanship, age, and many other factors.

Comparative Pipe Roughness Values

| Material | Manning's Coefficient | Hazen-Williams | Darcy-Weisbach Roughness Height | |
|---|---|---|---|---|
| | n | C | k (mm) | k (ft) |
| Asbestos Cement | 0.011 | 140 | 0.0015 | 0.000005 |
| Brass | 0.011 | 135 | 0.0015 | 0.000005 |
| Brick | 0.015 | 100 | 0.6 | 0.002 |
| Cast-iron, new | 0.012 | 130 | 0.26 | 0.00085 |
| Concrete: | | | | |
| Steel forms | 0.011 | 140 | 0.18 | 0.006 |
| Wooden forms | 0.015 | 120 | 0.6 | 0.002 |
| Centrifugally spun | 0.013 | 135 | 0.36 | 0.0012 |
| Copper | 0.011 | 135 | 0.0015 | 0.000005 |
| Corrugated metal | 0.022 | --- | 45 | 0.15 |
| Galvanized iron | 0.016 | 120 | 0.15 | 0.0005 |
| Glass | 0.011 | 140 | 0.0015 | 0.000005 |
| Lead | 0.011 | 135 | 0.0015 | 0.000005 |
| Plastic | 0.009 | 150 | 0.0015 | 0.000005 |
| Steel: | | | | |
| Coal-tar enamel | 0.010 | 148 | 0.0048 | 0.000016 |
| New unlined | 0.011 | 145 | 0.045 | 0.00015 |
| Riveted | 0.019 | 110 | 0.9 | 0.003 |
| Wood stave | 0.012 | 120 | 0.18 | 0.0006 |

Notes

# Appendix C

# Importing Loading Data

## C .1 Importing Loading Data Overview

SewerCAD has the capability of reading in loading data from a text file. This data includes:

- Pattern data
- Hydrograph data
- Multiple unit sanitary loads into the same hydraulic element
- Pattern loads

The format of the file format is quite flexible, and can be easily created by copying and pasting the data from the source application into a text editor such as *Notepad*.

The following help topics contain detailed information on the creation and import of the data file.

## C .2 Import Loading Data Dialog

From this dialog a file is selected for import, and import parameters are specified.

To import a file:

- Click on the **Browse** button in the **File Name** section to select the ASCII file to import.
- Select either to Add or Replace loads.
  - **Add Loads** - If this option is toggled the imported loads will be added to the list of loads already applied at the specified element.
  - **Replace Loads** - If this option is toggled the imported loads will overwrite any loads already applied at the specified element.
- Input data for the following options:
  - **Flow Units -** Select the units of the flow data in the import file.
  - **Pattern Type** - When importing patterns, choose whether the imported patterns are of type Continuous or Stepwise.
  - **Pattern Time Step** - Specify the time increment of the pattern data setup in the Constant Increment Pattern section.
  - **Hydrograph Time Step** - Specify the time increment of the hydrograph data setup in the Constant Increment Hydrographs section.

# C .3 Loading Data Text File Format

## C.3.1 ASCII Loading Data Format

SewerCAD has the ability to import various loading data from an ASCII text file. The file is divided into sections based on the type of information being imported.

A section begins with a header, which is the name of the section typed exactly in brackets. The header is followed by the data to be imported.

The following data sections can be imported:

- [Constant Increment Patterns]
- [Variable Increment Patterns]
- [Sanitary Pattern Loads]
- [Wet Pattern Loads]
- [Constant Increment Hydrographs]
- [Variable Increment Hydrographs]
- [Sanitary Hydrograph Loads]
- [Wet Hydrograph loads]
- [Sanitary Unit Loads]
- [Options]

You do not have to enter all these sections in the import file, just the sections you are interested in importing.

Comment lines can be placed anywhere in the file by putting a ";" at the beginning of the line.

## C.3.2 Constant Increment Patterns Section

From this section of the data import file you can import pattern data directly into the **Pattern Manager** for use in the model. In this section the pattern data should have a constant time increment. This time increment is established either in the **Pattern Time Step** field in the **Import Loading Data** dialog or in the Options section of the import file.

The format of the data is very flexible but must obey the following rules:

- The first row of the section contains *[Constant Increment Patterns]* and nothing else.
- The first column of each successive row of data must contain the name of the pattern for the associated data. For example *Pattern-1*, *Pattern 1* etc.
- If the pattern name contains a space it must be surrounded by quotation marks. For example *Pattern 1* should be typed as *"Pattern 1."*
- The following columns contain the actual pattern multipliers. They can be placed in as many columns following the pattern name as desired until you decide to go to the next row. So you could have a single column of multipliers or a single long row of multipliers or any combination of the two.

## C.3.3 Variable Increment Patterns Section

From this section of the data import file you can import pattern data directly into the **Pattern Manager** for use in the model. In this section the pattern data can have a variable time increment.

The format of the data is very flexible but must adhere to the following rules:

- The first row of the section contains *[Variable Increment Patterns]* and nothing else.
- The first column of each successive row of data must contain the name of the pattern for the associated data. For example *Pattern-1*, *Pattern1* etc.
- If the pattern name contains a space it must be surrounded by quotation marks. For example *Pattern 1* should be written as *"Pattern 1"*
- The following columns are entered in sets of two. The first value entered is the **Time from Start** and the second value is the **Multiplier**. These pairs can be placed in as many columns following the pattern name until you decide to go to the next row. So you could have two columns in addition to the pattern name for each row, or you could have a long row of *time multiplier time multiplier etc…* You could also, have any combination of the two.

## C.3.4 Sanitary Pattern Loads Section

From this section of the data import file you can import information pertaining to individual sanitary (dry-weather) pattern loads and apply them to hydraulic elements.

The format of the data must adhere to the following rules:

- The first row of the section contains *[Sanitary Pattern Loads]* and nothing else.
- The first column of each successive row contains the name of the hydraulic element to import the pattern data into. The elements must already exist in the model.

**Sanitary Pattern Loads can only be imported into manholes, wet wells, and pressure junctions.**

- The second column of each successive row contains the base flow associated with the pattern load. The unit of the flow is specified upon import in the **Import Loading Data** dialog.
- The third column of each successive row contains the name of the pattern that is to be applied to the pattern load. The patterns must already exist in the model or be imported in the same file as the pattern load information.
- If the names of the hydraulic elements, or the patterns contain a space then those names must be surrounded by quotation marks. For example, *Manhole 1* should be typed as *"Manhole 1"*, and *Pattern 1* should be written as *"Pattern 1"*.

## C.3.5 Wet Pattern Loads Section

From this section of the data import file you can import information pertaining to individual wet weather (infiltration and inflow) pattern loads and apply them to appropriate hydraulic elements.

The format of the data must adhere to the following rules:

- The first row of the section contains *[Wet Pattern Loads]* and nothing else.
- The first column of each successive row contains the name of the hydraulic element to import the pattern data into. The elements must already exist in the model.

**Wet Pattern Loads can only be imported into manholes, pressure junctions, wet wells, and into gravity pipes.**

- The second column of each successive row contains the base flow associated with the pattern load. The unit of the flow is specified upon import in the **Import Loading Data** dialog, or in the Options section of the import file.
- The third column of each successive row contains the name of the pattern that is to be applied to the pattern load. The patterns must already exist in the model or be imported in the same file as the pattern load information.
- If the names of the hydraulic elements, or the patterns contain a space then those names must be surrounded by quotation marks. For example, *Manhole 1* should be typed as *"Manhole 1"*, and *Pattern 1* should be written as *"Pattern 1"*.

## C.3.6 Constant Increment Hydrographs Section

From this section of the data import file you can import hydrograph data into the model to be applied to a hydraulic element. In this section the hydrograph data should have a constant time increment. This time increment is established either in the **Hydrograph Time Step** field in the **Import Loading Data** dialog or in the Options section of the import file.

The format of the data is very flexible but must adhere to the following rules:

- The first row of the section contains *[Constant Increment Hydrographs]* and nothing else.
- The first column of each successive row of data must contain the name of the pattern for the associated data. For example *Hydrograph-1*, *Hydrograph 1* etc.
- If the hydrograph name contains a space it must be surrounded by quotation marks. For example, *Hydrograph 1* should be written as *"Hydrograph 1."*
- The following columns of each successive row contain the actual flows. They can be placed in as many columns following the hydrograph name desired until you decide to go to the next row. So you could have a single column of flows, a single long row of flows or any combination of the two. The unit of the flow is specified upon import in the **Import Loading Data** dialog, or in the Options section of the import file.

**Hydrograph information is imported through this section. You specify where the hydrographs are applied through the Sanitary Hydrograph Loads and Wet Hydrograph Loads sections.**

## C.3.7 Variable Increment Hydrographs Section

From this section of the data import file you can import hydrograph data directly into the program for distribution to various hydraulic elements.

The format of the data is very flexible but must adhere to the following rules:

- The first row of the section contains *[Variable Increment Hydrographs]* and nothing else.
- The first column of each successive row of data must contain the name of the hydrograph for the associated data. For example *Hydrograph-1*, *Hydrograph 1* etc.
- If the hydrograph name contains a space it must be surrounded by quotation marks. For example *Hydrograph 1* should be written as *"Hydrograph 1."*
- The following columns are entered in sets of two. The first value entered is the **Time from Start** and the second value is the **Flow**. These pairs can be placed in as many columns following the hydrograph name until you decide to go to the next row. So you could have two columns in addition to the hydrograph name for each row, or you could have a long row of *time flow time flow etc…* You could also, have any combination of the two. The unit of the flow is specified upon import in the **Import**

**Loading Data** dialog, or in the Options section of the import file.

> **Hydrograph information is imported through this section. You specify where the hydrographs are applied through the Sanitary Hydrograph Loads and Wet Hydrograph Loads sections.**

### C.3.8 Sanitary Hydrograph Loads Section

From this section of the data import file you specify to what hydraulic elements the hydrographs entered in the Variable Increment Hydrographs and Constant Increment Hydrographs sections of the import file are applied.

The format of the data must adhere to the following rules:

- The first row of the section contains *[Sanitary Hydrograph Loads]* and nothing else.
- The first column of each successive row contains the name of the hydraulic element to import the pattern data into. The elements must already exist in the model.

> **Sanitary Hydrograph Loads can only be imported into manholes, wet wells, and pressure junctions.**

- The second column of each successive row contains the name of the hydrograph that is to be applied to the hydraulic element. The hydrographs referred to in this section must be imported in the same file as the hydrograph load information.
- If the names of the hydraulic elements or the hydrographs contain a space then those names must be surrounded by quotation marks. For example, *Manhole 1* should be written as *"Manhole 1"*, and *Hydrograph 1* should be written as *"Hydrograph 1"*.

### C.3.9 Wet Hydrograph Loads Section

From this section of the data import file you can attribute the hydrographs entered in the Variable Increment Hydrographs and Constant Increment Hydrographs sections of the import file to different hydraulic elements.

The format of the data must adhere to the following rules:

- The first row of the section contains *[Wet Hydrograph Loads]* and nothing else.
- The first column of each successive row contains the name of the hydraulic element to import the hydrograph data into. The elements must already exist in the model.

> **Wet Hydrograph Loads can only be imported into manholes pressure junctions, and gravity pipes.**

- The second column of each successive row contains the name of the hydrograph that is to be applied to the hydraulic element. The hydrographs referred to in this section must be imported in the same file as the hydrograph load information.
- If the names of the hydraulic elements, or the hydrograph contain a space then those names must be surrounded by quotation marks. For example, *Manhole 1* should be written as *"Manhole 1"*, and *Hydrograph 1* should be written as *"Hydrograph 1"*.

## C.3.10 Sanitary Unit Loads Section

From this section of the data import file you can import specific unit loads and attribute them to specified hydraulic elements.

The format of the data must adhere to the following rules:

- The first row of the section contains *[Sanitary Unit Loads]* and nothing else.
- The first column of each successive row contains the name of the hydraulic element to import the sanitary unit load into.

**Sanitary Unit Loads can only be imported into manholes, wet wells, and pressure junctions.**

- The second row of each successive column contains the name of the **Unit Sanitary (Dry Weather) Load**. For example, *Airport, Home (Residential), etc…* The **Unit Sanitary Load** must already be specified in the Engineering Libraries for it to be correctly applied.
- The third row of each successive column contains a number, the Unit Load count.
- If the names of the hydraulic elements, or the **Unit Sanitary Load** contain a space then the name must be surrounded by quotation marks. For example, *Manhole 1* should be written as *"Manhole 1"*, and *Bar per Customer* should be written as *"Bar per Customer"*.

## C.3.11 Options Section

From this section you can initialize the option parameters on the **Import Loading Data** dialog. This initialization occurs upon selection of the import file in the dialog.

You can read in data for the following parameters:

- **Pattern Time Step** - set to a positive numerical value.
- **Hydrograph Time Step** - set to a positive numerical value.
- **Flow Units** - set to one of the units selectable from the **Flow Units** field in the **Import Loading Data** dialog.

**If the flow unit contains a superscript such as $m^3/s$, you can add the superscript by using Character Map, which is a standard Windows application, which can be opened through the Accessories directory in the Start menu. Otherwise simply select the unit in the Flow Units field on the Import Loading Data dialog.**

- Pattern Type - set to either Continuous or Stepwise.

The format of the data must adhere to the following rules:

- The first row of the section contains *[Options]* and nothing else.
- Each successive row of the section (up to four) contains the name of the parameter typed exactly as presented above; followed by a space; followed by an equal sign; followed by a space, and then the value of the parameter.

**The Options section is not required for the data import to run successfully, and beyond that you only have of specify parameters for the options that you require.**

**The Options section only initializes the values on the Import Loading Data dialog. If you change the values in the dialog after specifying the import file, the values in the dialog will be used.**

## C.3.12 ASCII Loading Data Example

This help topic contains a sample ASCII Loading File, and contains sample data for all the different sections that are capable of import. Note the variability in the way columns and rows can be setup, and how items can be named.

```
[Constant Increment Patterns]
Pattern-1  1.0 2.0 3.0
Pattern-1  3.0 2.0 1.0

[Variable Increment Patterns]
; This is information about
;  this pattern  set on two comment lines.
"Pattern 2"   1.5 1.0   2.5 2.0   3.0 3.0
"Pattern 2"   4.0 2.0   5.5 1.0

[Sanitary Pattern Loads]
MH-1      100.0     "Pattern-1"
MH-1      200.0     "Pattern 2"

[Wet Pattern Loads]
"MH-1" 100.0       Pattern-1
"MH-1" 200.0      "Pattern 2"

[Constant Increment Hydrographs]
"Hydrograph 1"     1.0        2.0         3.0
"Hydrograph 1"     3.0        2.0         1.0

[Variable Increment Hydrographs]
Hydrograph-2    0.25    0.501302
Hydrograph-2    0.50    1.002604
Hydrograph-2    1.00    2.005208
Hydrograph-2    1.50    3.007813
Hydrograph-2    2.00    3.007813
Hydrograph-2    2.50    2.005208
Hydrograph-2    3.00    1.002604

[Sanitary Hydrograph Loads]
MH-1      "Hydrograph 1"
MH-1       Hydrograph-2
J-1       "Hydrograph 1"

[Wet Hydrograph Loads]
"MH-1"  "Hydrograph 1"
"MH-1"    Hydrograph-2
"P-1"      “Hydrograph 1”

[Sanitary Unit Loads]
"MH-1"  "Airport"                  53.0
"MH-1"  "Apartment"                500.0
"J-1"   "Airport"                  53.0

[Options]
Pattern Time Step = 0.25
Hydrograph Time Step = 0.5
Flow Units = cfs
Pattern Type = Continuous
```

Notes

# Appendix D

# Scenario Management Reference Guide

## D .1 Overview

Haestad Methods' scenario management feature can dramatically increase your productivity in the "W*hat If?"* areas of modeling, including calibration, operations analysis, and planning.

By investing a little time now to understand scenario management, you can avoid unnecessary editing and data duplication. Take advantage of scenario management to get a lot more out of your model, with much less work and expense.

In contrast to the old methods of scenario management (editing or copying data), automated scenario management using inheritance gives you significant advantages:

- A single project file makes it possible to generate an unlimited number of "*What If?"* conditions without becoming overwhelmed with numerous modeling files and separate results.
- Because the software maintains the data for all the scenarios in a single project, it can provide you with powerful automated tools for directly comparing scenario results. Any set of results is immediately available at any time.
- The Scenario / Alternative relationship empowers you to mix and match groups of data from existing scenarios without having to re-declare any data.
- With inheritance, you do not have to re-enter data if it remains unchanged in a new alternative or scenario, avoiding redundant copies of the same data. Inheritance also enables you to correct a data input error in a parent scenario and automatically update the corrected attribute in all child scenarios.

These advantages, while obvious, may not seem compelling for small projects. It is as projects grow to hundreds or thousands of network elements that the advantages of true scenario inheritance become clear. On a large project, being able to maintain a collection of base and modified alternatives accurately and efficiently can be the difference between evaluating optional improvements and being forced to ignore them.

## D .2 About this Guide

The depth of scenario management as implemented by Haestad Methods is probably far beyond what you have ever seen before. With that in mind, this guide is intended as an introduction to the philosophy and terminology upon which scenario management is based.

**This is not intended as a step-by-step guide to using the software.** If you are a moderately experienced Windows software user, you should have no difficulty learning and exploring the scenario management interface.

Excellent tutorials and context-sensitive on-line help are also available within the software itself. These learning tools will prove to be of tremendous assistance to you for all aspects of the software, and should certainly not be ignored if you are having difficulty. For more information, just click the **Help** button, which is available from anywhere within the program. In addition, contact Haestad Methods for the schedule of the workshops that are held around the country.

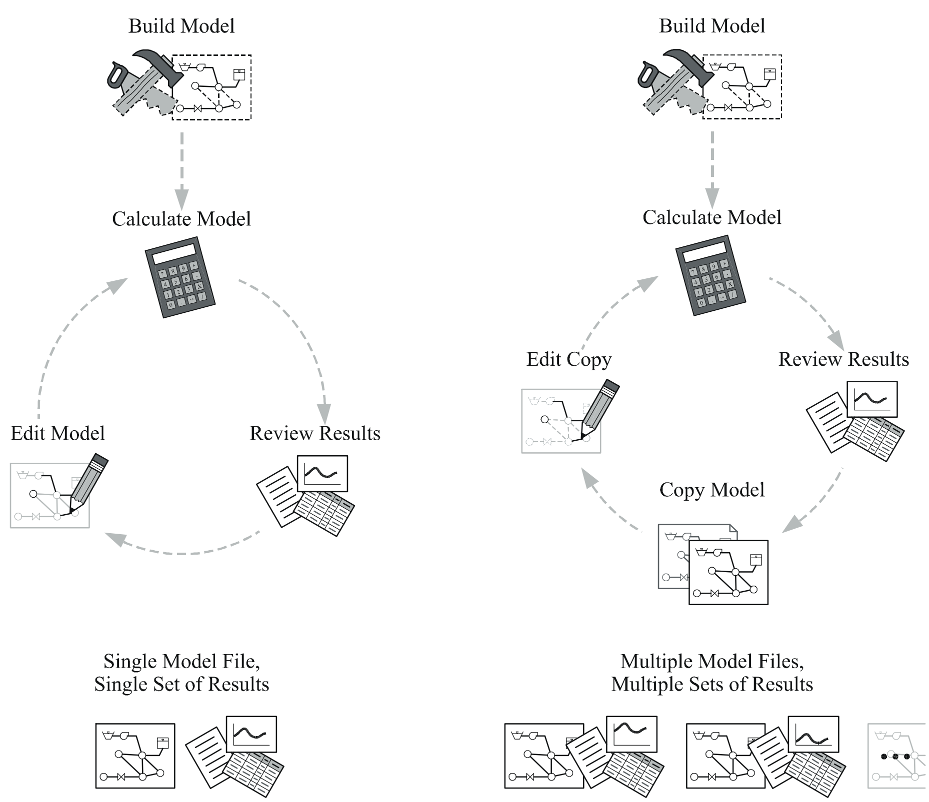
## D .3 Before Haestad Methods: Distributed Scenarios

Let us begin by understanding the approaches that have historically been used to attempt *"What If?"* analyses. Traditionally, there have only been two possible ways of analyzing the effects of change on a software model:

- Change the model, recalculate, and review the results
- Create a copy of the model, edit that copy, calculate, and review the results

Although either of these methods may be adequate for a relatively small system, the data duplication, editing, and re-editing becomes very time-consuming and error-prone as the size of the system - and the number of possible conditions - increase. Additionally, comparing conditions requires manual data manipulation, because all output must be stored in physically separate data files.

Before Haestad Methods: Distributed Scenarios



## D .4 With Haestad Methods: Self-Contained Scenarios

Effective scenario management tools need to meet these objectives:

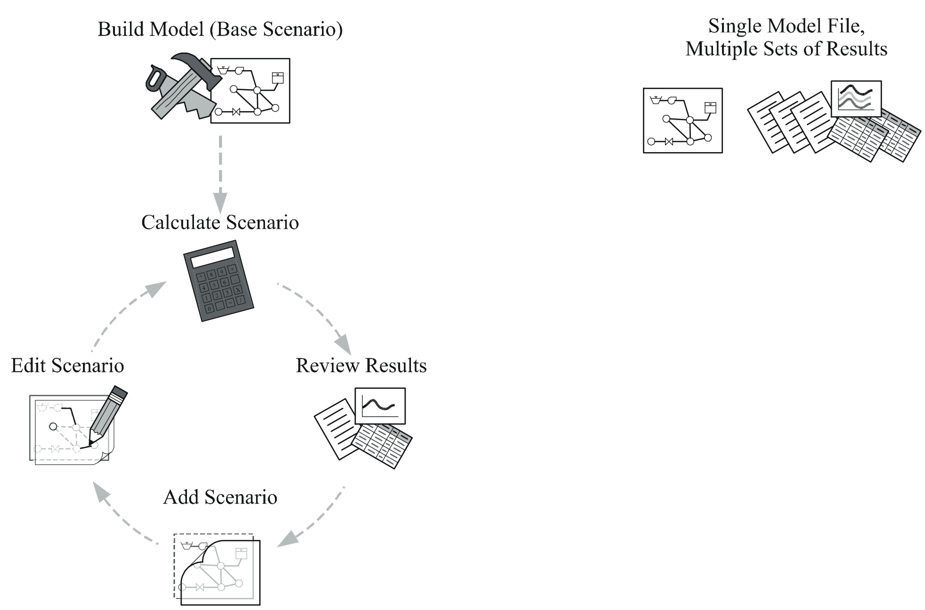
- Minimize the number of project files the modeler needs to maintain (one, ideally).
- Maximize the usefulness of scenarios through easy access to things such as input and output data, and direct comparisons.
- Maximize the number of scenarios you can simulate by mixing and matching data from existing scenarios (data reuse)
- Minimize the amount of data that needs to be duplicated to consider conditions that have a lot in common

The scenario management feature developed by Haestad Methods successfully meets all of these objectives. A single project file enables you to generate an unlimited number of "W*hat If?"* conditions, edit only the data that needs to be changed, and quickly generate direct comparisons of input and results for desired scenarios.

## D .5 The Scenario Cycle

The process of working with scenarios is similar to the process of manually copying and editing data, but without the disadvantages of data duplication and troublesome file management. This process allows you to cycle through any number of changes to the model, without fear of overwriting critical data or duplicating important information. Of course, it is possible to directly change data for any scenario, but an "audit trail" of scenarios can be useful for retracing the steps of a calibration series or for understanding a group of master plan updates.

With Haestad Methods: Self-contained Scenarios



# D .6 Scenario Anatomy: Attributes and Alternatives

Before we explore scenario management further, a few key terms should be defined:

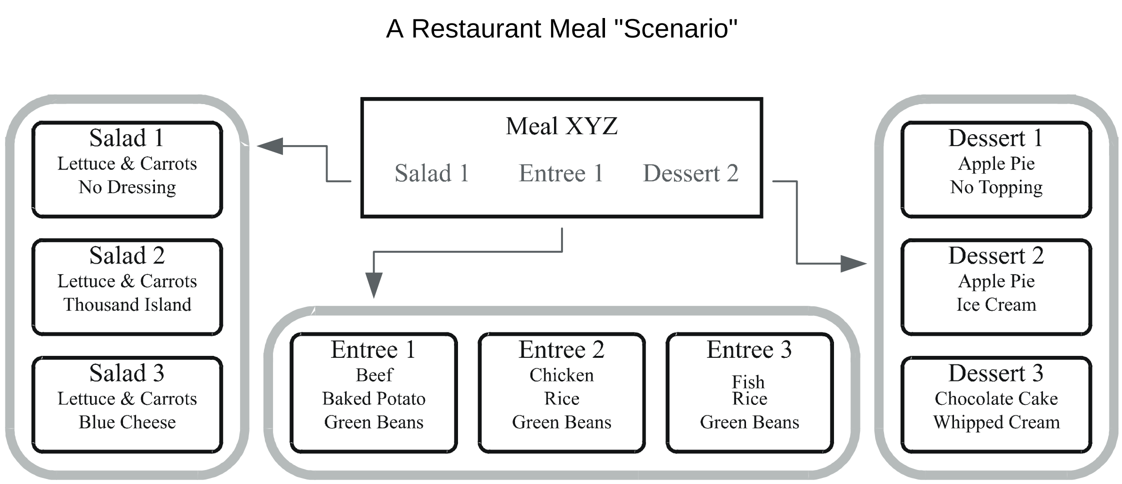
**Attribute** - An attribute is a fundamental property of an object, and is often a single numeric quantity. For example, the attributes of a pipe include diameter, length, and roughness.

**Alternative** - An alternative holds a family of related attributes so pieces of data that you are most likely to change together are grouped for easy referencing and editing. For example, a physical properties alternative groups physical data for the network's elements, such as elevations, sizes, and roughness coefficients.

**Scenario** - A scenario has a list of referenced alternatives (which hold the attributes), and combines these alternatives to form an overall set of system conditions that can be analyzed. This referencing of alternatives enables you to easily generate system conditions that mix and match groups of data that have been previously created. Note that scenarios do not actually hold any attribute data - the referenced alternatives do.

# D .7 A Familiar Parallel

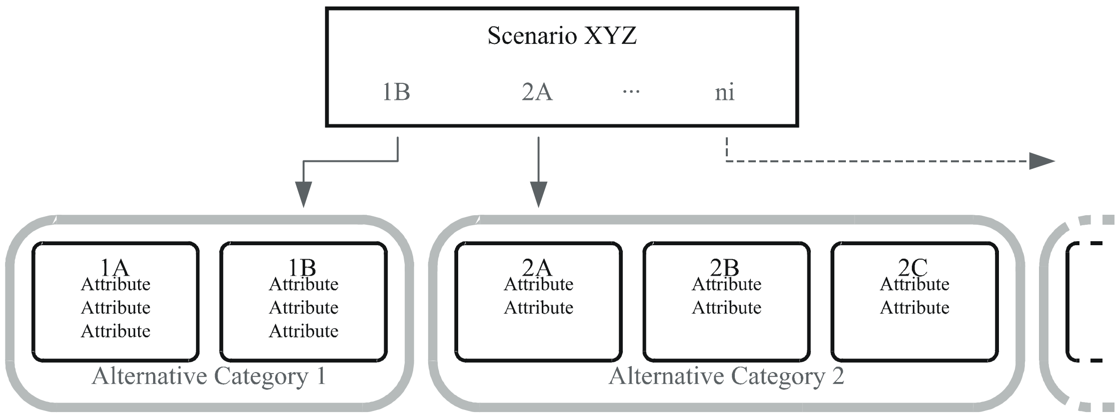
Although the structure of scenarios may seem a bit difficult at first, anyone who has eaten at a restaurant should be able to relate fairly easily. A meal (scenario) is comprised of several courses (alternatives), which might include a salad, an entrée, and a dessert. Each course has its own attributes. For example, the entrée may have a meat, a vegetable, and a starch. Examining the choices, we could present a menu as in the following figure:




The restaurant does not have to create a new recipe for every possible meal (combination of courses) that could be ordered. They can just assemble any meal based on what the customer orders for each alternative course. Salad 1, Entrée 1, and Dessert 2 might then be combined to define a complete meal.

Generalizing this concept, we see that any scenario simply references one alternative from each category to create a "big picture" that can be analyzed. Note that different types of alternatives may have different numbers and types of attributes, and any category can have an unlimited number of alternatives to choose from.

Generic Scenario Anatomy




## D .8 Scenario Behavior: Inheritance

The separation of scenarios into distinct alternatives (groups of data) meets one of the basic goals of scenario management: maximizing the number of scenarios you can develop by mixing and matching existing alternatives. Two other primary goals have also been addressed: a single project file is used, and easy access to input data and calculated results is provided in numerous formats through the intuitive graphical interface.

But what about the other objective: minimizing the amount of data that needs to be duplicated to consider conditions that have a lot of common input? Surely an entire set of pipe diameters should not be re-specified if only one or two change?

The solution is a familiar concept to most people: ***inheritance***.

In the natural world, a child inherits characteristics from a parent. This may include such traits as eye-color, hair color, and bone structure. There are two significant differences between the genetic inheritance that most of us know and the way inheritance is implemented in software:

- Overriding inheritance
- Dynamic inheritance

## D .9 Overriding Inheritance

Overriding inheritance is the software equivalent of cosmetics. A child can override inherited characteristics at any time by specifying a new value for that characteristic. These overriding values do not affect the parent, and are therefore considered "local" to the child. Local values can also be removed at any time, reverting the characteristic to its inherited state. The child has no choice in the value of his inherited attributes, only in local attributes.

For example, suppose a child has inherited the attribute of blue eyes from his parent. Now the child puts on a pair of green- tinted contact lenses to hide his natural eye color. When the contact lenses are on, we say his natural eye color is "overridden" locally, and his eye color is green. When the child removes the tinted lenses, his eye color instantly reverts to blue, as inherited from his parent.

## D .10 Dynamic Inheritance

Dynamic inheritance does not have a parallel in the genetic world. When a parent's characteristic is changed, existing children also reflect the change. Using the eye-color example, this would be the equivalent of the parent changing eye color from blue to brown, and the children's eyes instantly inheriting the brown color also. Of course, if the child has already overridden a characteristic locally, as with the green lenses, his eyes will remain green until the lenses are removed. At this point, his eye color will revert to the inherited color, now brown.

This dynamic inheritance has remarkable benefits for applying wide-scale changes to a model, fixing an error, and so on. If rippling changes are **not** desired, the child can override all of the parent's values, or a copy of the parent can be made instead of a child.

## D .11 When are Values Local, and When are They Inherited?

Any **changes** that are made to the model belong to the currently active scenario and the alternatives that it references. If the alternatives happen to have children, those children will also inherit the changes unless they have specifically overridden that attribute. The following figure demonstrates the effects of a change to a mid-level alternative. Inherited values are shown as gray text, local values are shown as black text.

A Mid-level Hierarchy Alternative Change

| Alternative Hierarchy | Value |
|---|---|
| Condition 1 | 100 |
| └ Condition 2 | 100 |
| └ Condition 3 | 100 |
| └ Condition 4 | 100 |
| └ Condition 5 | 150 |

→ *Change Condition 3 Value to 200* →

| Alternative Hierarchy | Value |
|---|---|
| Condition 1 | 100 |
| └ Condition 2 | 100 |
| └ Condition 3 | *200* |
| └ Condition 4 | 200 |
| └ Condition 5 | 150 |

## D .12 Minimizing Effort through Attribute Inheritance

Inheritance has an application every time you hear the phrase "just like *x* except for *y*". Rather than specifying all of the data from *x* again to form this new condition, we can simply create a child from *x* and change *y* appropriately. Now we have both conditions, with no duplicated effort.

We can even apply this inheritance to our restaurant analogy as follows. Inherited values are shown as gray text, local values are shown as black text.

| Salad Alternative Hierarchy | Attribute: Vegetables | Attribute: Dressing |
|---|---|---|
| Salad 1 | Lettuce & Carrots | No Dressing |
| ├ Salad 2 | Lettuce & Carrots | Thousand Island |
| └ Salad 3 | Lettuce & Carrots | Blue Cheese |

- "Salad 2 is just like Salad 1, except for the dressing."
- "Salad 3 is just like Salad 1, except for the dressing."

> **Salad 3 could inherit from Salad 2, if we prefer: "Salad 3 is just like Salad 2, except for the dressing."**

| Entree Alternative Hierarchy | Attribute: Meat | Attribute: Starch | Attribute: Vegetable |
|---|---|---|---|
| Entree 1 | Beef | Baked Potato | Green Beans |
| └ Entree 2 | Chicken | Rice | Green Beans |
| └ Entree 3 | Fish | Rice | Green Beans |

- "Entrée 2 is just like Entrée 1, except for the meat and the starch."
- "Entrée 3 is just like Entrée 2, except for the meat."

**If the vegetable of the day changes (say from green beans to peas), only Entrée 1 needs to be updated, and the other entrées will automatically inherit the vegetable attribute of "Peas" instead of "Green Beans".**

| Dessert Alternative Hierarchy | Attribute: Bakery Item | Attribute: Topping |
|---|---|---|
| Dessert 1 | Apple Pie | No Topping |
| └ Dessert 2 | Apple Pie | Ice Cream |
| Dessert 3 | Chocolate Cake | Whipped Cream |

- "Dessert 2 is just like Dessert 1, except for the topping."

**Dessert 3 has nothing in common with the other desserts, so it can be created as a "root" or "base" alternative. It does not inherit its attribute data from any other alternative.**

## D .13 Minimizing Effort through Scenario Inheritance

Just as a child alternative can inherit attributes from its parent, a child scenario can inherit which alternatives it references from its parent. This is essentially still the phrase "just like *x* except for *y*", but on a larger scale.

Carrying through on our meal example, consider a situation where you go out to dinner with three friends. The first friend places his order, and the second friend orders the same thing except for the dessert. The third friend orders something totally different, and you order the same meal as hers except for the salad.

The four meal "scenarios" could then be presented as follows (inherited values are shown as gray text, local values are shown as black text):

| Meal Scenario Hierarchy | Salad Alternative | Entree Alternative | Dessert Alternative |
|---|---|---|---|
| Meal 1 | Salad 1 | Entree 2 | Dessert 3 |
| └ Meal 2 | Salad 1 | Entree 2 | Dessert 1 |
| Meal 3 | Salad 3 | Entree 3 | Dessert 2 |
| └ Meal 4 | Salad 2 | Entree 3 | Dessert 2 |

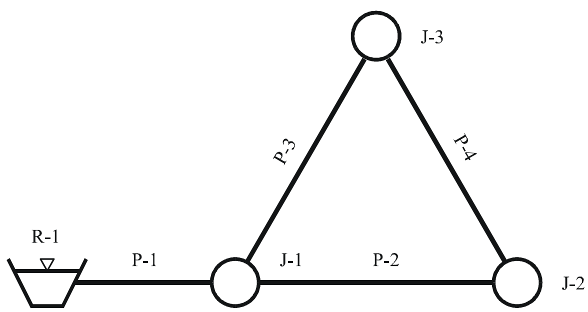
- "Meal 2 is just like Meal 1, except for the dessert." The salad and entrée alternatives are inherited from Meal 1.
- "Meal 3 is nothing like Meal 1 or Meal 2." A totally new "base" or "root" is created.

- "Meal 4 is just like Meal 3, except for the salad." The entrée and dessert alternatives are inherited from Meal 3.

## D .14 A Water Distribution Example

Let us consider a fairly simple water distribution system: a single reservoir supplies water by gravity to three junction nodes.

Example Water Distribution System




Although true water distribution scenarios include such alternative categories as initial settings, operational controls, water quality, and fire flow, we are going to focus on the two most commonly changed sets of alternatives: demands and physical properties. Within these alternatives, we are going to concentrate on junction baseline demands and pipe diameters.

## D .15 Building the Model (Average Day Conditions)

During model construction, probably only one alternative from each category is going to be considered. This model is built with average demand calculations and preliminary pipe diameter estimates. At this point we can name our scenario and alternatives, and the hierarchies look like the following (showing only the items of interest):

| Demand Alternative Hierarchy | J-1 | J-2 | J-3 |
|---|---|---|---|
| *Average Day* | *100 gpm* | *500 gpm* | *100 gpm* |

| Physical Alternative Hierarchy | P-1 | P-2 | P-3 | P-4 |
|---|---|---|---|---|
| *Preliminary Pipes* | *8 inches* | *6 inches* | *6 inches* | *6 inches* |

| Scenario Hierarchy | Demand Alternative | Physical Alternative |
|---|---|---|
| *Avg. Day* | *Average Day* | *Preliminary Pipes* |

## D .16 Analyzing Different Demands (Maximum Day Conditions)

In our example, the local planning board also requires analysis of maximum day demands, so a new demand alternative is required. No variation in demand is expected at J-2, which is an industrial site. As a result, the new demand alternative can inherit J-2's demand from "Average Day" while the other two demands are overridden.

| Demand Alternative Hierarchy | J-1 | J-2 | J-3 |
|---|---|---|---|
| Average Day | 100 gpm | 500 gpm | 100 gpm |
| └ *Maximum Day* | *200 gpm* | 500 gpm | *200 gpm* |

Now we can create a child scenario from "Average Day" that inherits the physical alternative, but overrides the selected demand alternative. As a result, we get the following scenario hierarchy:

| Scenario Hierarchy | Demand Alternative | Physical Alternative |
|---|---|---|
| Avg. Day | Average Day | Preliminary Pipes |
| └ *Max. Day* | *Maximum Day* | Preliminary Pipes |

Since no physical data (pipe diameters) have been changed, the physical alternative hierarchy remains the same as before.

## D .17 Another Set of Demands (Peak Hour Conditions)

Based on pressure requirements, the system is adequate to supply maximum day demands. Another local regulation requires analysis of peak hour demands, with slightly lower allowable pressures. Since the peak hour demands also share the industrial load from the "Average Day" condition, "Peak Hour" can be inherited from "Average Day". In this instance, "Peak Hour" could inherit just as easily from "Maximum Day".

| Demand Alternative Hierarchy | J-1 | J-2 | J-3 |
|---|---|---|---|
| Average Day | 100 gpm | 500 gpm | 100 gpm |
| ├ Maximum Day | 200 gpm | 500 gpm | 200 gpm |
| └ *Peak Hour* | *250 gpm* | 500 gpm | *250 gpm* |

Another scenario is also created to reference these new demands, as shown below:

| Scenario Hierarchy | Demand Alternative | Physical Alternative |
|---|---|---|
| Avg. Day | Average Day | Preliminary Pipes |
| ├ Max. Day | Maximum Day | Preliminary Pipes |
| └ *Peak* | *Peak Hour* | Preliminary Pipes |

Note again that we did not change any physical data, so the physical alternatives remain the same.

## D .18 Correcting an Error

This analysis results in acceptable pressures, until it is discovered that the industrial demand is not actually 500 gpm - it is 1,500 gpm! Because of the inheritance within the demand alternatives, however, only the "Average Day" demand for J-2 needs to be updated. The changes will ripple through to the children. After the single change is made, the demand hierarchy is as follows:

| Demand Alternative Hierarchy | J-1 | J-2 | J-3 |
|---|---|---|---|
| Average Day | 100 gpm | *1,500 gpm* | 100 gpm |
| ├ Maximum Day | 200 gpm | 1,500 gpm | 200 gpm |
| └ Peak Hour | 250 gpm | 1,500 gpm | 250 gpm |

Notice that no changes need to be made to the scenarios to reflect these corrections. The three scenarios can now be calculated as a batch to update the results.

When these results are reviewed, it is determined that the system does **not** have the ability to adequately supply the system as it was originally thought. The pressure at J-2 is too low under peak hour demand conditions.

## D .19 Analyzing Improvement Suggestions

To counter the headloss from the increased demand load, two possible improvements are suggested:

- A much larger diameter is proposed for P-1 (the pipe from the reservoir). This physical alternative is created as a child of the "Preliminary Pipes" alternative, inheriting all the diameters except P-1's, which is overridden.
- Slightly larger diameters are proposed for all pipes. Since there are no commonalities between this recommendation and either of the other physical alternatives, this can be created as a base (root) alternative.

These changes are then incorporated to arrive at the following hierarchies:

| Physical Alternative Hierarchy | P-1 | P-2 | P-3 | P-4 |
|---|---|---|---|---|
| Preliminary Pipes | 8 inches | 6 inches | 6 inches | 6 inches |
| └ *Larger P-1* | *18 inches* | 6 inches | 6 inches | 6 inches |
| *Larger All Pipes* | *12 inches* | *12 inches* | *12 inches* | *12 inches* |

| Scenario Hierarchy | Demand Alternative | Physical Alternative |
|---|---|---|
| Avg. Day | Average Day | Preliminary Pipes |
| ├ Max. Day | Maximum Day | Preliminary Pipes |
| └ Peak | Peak Hour | Preliminary Pipes |
| ├ *Peak, Big P-1* | Peak Hour | *Larger P-1* |
| └ *Peak, All Big Pipes* | Peak Hour | *Larger All Pipes* |

This time, the demand alternative hierarchy remains the same since no demands were changed. The two new scenarios ("Peak, Big P-1", "Peak, All Big Pipes") can be batch run to provide results for these proposed improvements.

Next, features like Scenario Comparison Annotation (from the Scenario Manager) and comparison Graphs (for extended period simulations, from the element editor dialogs) can be used to directly determine which proposal results in the most improved pressures.

## D .20 Finalizing the Project

It is decided that enlarging P-1 is the optimum solution, so new scenarios are created to check the results for average day and maximum day demands. Notice that this step does not require handling any new data. All of the information we want to model is present in the alternatives we already have!

| Scenario Hierarchy | Demand Alternative | Physical Alternative |
|---|---|---|
| Avg. Day | Average Day | Preliminary Pipes |
| ├ Max. Day | Maximum Day | Preliminary Pipes |
| │ └ *Max. Day, Big P-1* | Maximum Day | *Larger P-1* |
| ├ Peak | Peak Hour | Preliminary Pipes |
| │ ├ Peak, Big P-1 | Peak Hour | Larger P-1 |
| │ └ Peak, All Big Pipes | Peak Hour | Larger All Pipes |
| └ *Avg. Day, Big P-1* | Average Day | *Larger P-1* |

Also note that it would be equally effective in this case to inherit the "Avg. Day, Big P-1" scenario from "Avg. Day" (changing the physical alternative) or to inherit from "Peak, Big P-1" (changing the demand alternative). Likewise, "Max. Day, Big P-1" could inherit from either "Max. Day" or "Peak, Big P-1".

Neither the demand nor physical alternative hierarchies were changed in order to run the last set of scenarios, so they remain as they were.

| Demand Alternative Hierarchy | J-1 | J-2 | J-3 |
|---|---|---|---|
| Average Day | 100 gpm | 1,500 gpm | 100 gpm |
| ├ Maximum Day | 200 gpm | 1,500 gpm | 200 gpm |
| └ Peak Hour | 250 gpm | 1,500 gpm | 250 gpm |

| Physical Alternative Hierarchy | P-1 | P-2 | P-3 | P-4 |
|---|---|---|---|---|
| Preliminary Pipes | 8 inches | 6 inches | 6 inches | 6 inches |
| └ Larger P-1 | 18 inches | 6 inches | 6 inches | 6 inches |
| Larger All Pipes | 12 inches | 12 inches | 12 inches | 12 inches |

## D .21 Summary

In contrast to the old methods of scenario management (editing or copying data), automated scenario management using inheritance gives you significant advantages:

- A single project file makes it possible to generate an unlimited number of "*What If?*" conditions without becoming overwhelmed with numerous modeling files and separate results.
- Because the software maintains the data for all the scenarios in a single project, it can provide you with powerful automated tools for directly comparing scenario results. Any set of results is immediately available at any time.
- The Scenario / Alternative relationship empowers you to mix and match groups of data from existing scenarios without having to re-declare any data.
- With inheritance, you do not have to re-enter data if it remains unchanged in a new alternative or scenario, avoiding redundant copies of the same data. Inheritance also enables you to correct a data input error in a parent scenario and automatically update the corrected attribute in all child scenarios.

These advantages, while obvious, may not seem compelling for small projects. It is as projects grow to hundreds or thousands of network elements that the advantages of true scenario inheritance become clear. On a large project, being able to maintain a collection of base and modified alternatives accurately and efficiently can be the difference between evaluating optional improvements and being forced to ignore them.

## D .22 Conclusion

These are the fundamental concepts behind the architecture of Haestad Methods' scenario management. To learn more about actually using scenario management in Haestad Methods software, start by running the scenario management tutorial from the Help menu or from within the scenario manager itself. Then load one of the SAMPLE projects and explore the scenarios defined there. For context-sensitive help, press **F1** or the **Help** button any time there is a screen or field that puzzles you.

Haestad Methods' scenario management feature gives you a powerful tool for modeling real-world engineering scenarios when analyzing system response to different demands, reviewing the impacts of future growth, and iterating to find the least expensive design. That means you will be able to finish your projects faster, spend less money, and improve your bottom line.

# Appendix E

# Haestad Methods Software

## E .1 Overview

Haestad Methods offers software solutions to civil engineers throughout the world for analyzing, modeling, and designing all sorts of hydrologic and hydraulic systems, from municipal water and sewer systems to stormwater ponds, open channels, and more. With point-and-click data entry, flexible units, and report-quality output, Haestad Methods is the ultimate source for your modeling needs.

In addition to the ability to run in Stand-Alone mode with a CAD-like interface, three of our products - WaterCAD, StormCAD and SewerCAD - can be totally integrated within AutoCAD. These three programs also share numerous powerful features, such as scenario management, unlimited undo/redo, customizable tables for editing and reporting, customizable GIS, database and spreadsheet connection, and annotation.

Be sure to contact us or visit our web site at **www.haestad.com** to find out about our latest software, books, training, and open houses.

## E .2 WaterCAD

WaterCAD is the definitive model for complex pressurized pipe networks, such as municipal water distribution systems. You can use WaterCAD to perform a variety of functions, including steady-state and extended-period simulations of pressure networks with pumps, tanks, control valves, and more.

WaterCAD's abilities also extend into public safety and long-term planning issues, with extensive water quality features, automated fire protection analyses, comprehensive scenario management, and enterprise-wide data sharing faculties.

WaterCAD is available with your choice of a Stand-Alone graphical user interface, an AutoCAD integrated interface, or an ArcView or ArcInfo integrated interface.

## E .3 SewerCAD

SewerCAD is a powerful design and analysis tool for modeling sanitary sewage collection and pumping systems. With SewerCAD, you can develop and compute sanitary loads, tracking and combining loads from dry-weather and wet-weather sources. You can also simulate the hydraulic response of the entire system (gravity collection and pressure force mains), observe the effects of overflows and diversions, and even automatically design selected portions of the system. Output covers everything from customizable tables and detailed reports to plan and profile sheets.

SewerCAD can be run in a Stand-Alone graphical user interface, an AutoCAD integrated interface, or an ArcView or ArcInfo integrated interface.

## E .4 StormCAD

StormCAD is a highly-efficient model for the design and analysis of storm sewer collection systems. From graphical layout and intelligent network connectivity to flexible reports and profiles, StormCAD covers all aspects of storm sewer modeling.

Surface inlet networks are independent of pipe connectivity, and inlet hydraulics conform to FHWA HEC-22 methodologies. Gradually varied flow algorithms and a variety of popular junction loss methods are the foundation of StormCAD's robust gravity piping computations, which handle everything from surcharged pipes and diversions to hydraulic jumps.

StormCAD is available with your choice of a Stand-Alone graphical user interface, an AutoCAD integrated interface, or an ArcView or ArcInfo integrated interface.

## E .5 PondPack

PondPack is a comprehensive, Windows-based hydrologic modeling program that analyzes a tremendous range of situations, from simple sites to complex networked watersheds. The program analyzes pre- and post-developed watershed conditions, and estimates required storage ponds. PondPack performs interconnected pond routing, and also computes outlet rating curves with tailwater effects, multiple outfalls, pond infiltration, and pond detention times.

PondPack builds customized reports organized by categories, and automatically creates section and page numbers, tables of contents, and indexes. You can quickly create an executive summary for an entire watershed, or build an elaborate drainage report showing any or all report items. Graphical displays, such as watershed diagrams, rainfall curves, and hydrographs are fully compatible with other Windows's software, such as AutoCAD.

## E .6 CulvertMaster

CulvertMaster helps engineers design new culverts and analyze existing culvert hydraulics, from single barrel crossings to complex multi-barrel culverts with roadway overtopping. CulvertMaster computations use FHWA HDS-5 methodologies, and allow you to solve for whatever hydraulic variables you don't know, such as culvert size, peak discharge, and headwater elevation. Output capabilities include comprehensive detailed reports, rating tables, and performance curves.

## E .7 FlowMaster

FlowMaster is an efficient program for the design and analysis of a wide variety of hydraulic elements, such as pressure pipes, open channels, weirs, orifices, and inlets. FlowMaster's "Hydraulics Toolbox" can create rating tables and performance curves for any variables, using popular friction methods. Inlet calculations follow the latest FHWA guidelines, and irregular section roughness can be weighted based on any popular techniques.

# Glossary

**AASHTO Headloss Method -** Automatic Computation of the junction loss based on the geometry of the junction, which accounts for bend loss, based on the upstream pipes angle, as defined in the AASHTO *Model Drainage Manual* (1991).

**AASHTO Shaping Method -** Specifies whether the junction bottom is designed with a partial diameter shaping or not. See Headloss - AASHTO Method.

**Absolute Headloss -** A user specified headloss - See Absolute Headloss Method.

**Absolute Roughness -** Average height of roughness particles in the channel.

**Additional Infiltration -** Lumped infiltration amount.

**Average Velocity -** Average water velocity in the pipe. The following methods are available for calculating the average velocity: Uniform Flow, Full Flow, Simple Average and Weighted Average. Refer to the theory chapter for explanations.

**Base Load -** Average local sanitary load calculated by multiplying the Unit Base Load by the Loading Unit Count for every Unit Sanitary Load category (for instance Apartment or Airport) at a loading point.

**Bend Loss Coefficient -** Bend Loss Coefficient K used in AASHTO equation for structure headloss calculations.

**Bend Angle -**Angle between the pipe and the downstream pipe, measured as a deflection angle. Note that this angle is used in the HEC22 and AASHTO junction loss methods.

**Bolted Cover -** Indicates whether a manhole is bolted. If the manhole is bolted, then the hydraulic grade line is not reset to the rim elevation at the downstream end of the upstream pipe(s) in the case of a flooding situation (the calculated HGL being higher than the rim elevation).

**Calculated Hydraulic Grade -** Hydraulic Grade in the wet well calculated as explained in the Hydraulic Transition from Gravity to Pressure Network section in the Theory chapter.

**Cancel Button -** When you click on this button, it cancels the command you chose and closes the dialog.

**Capacity -** The effective carrying ability of a drainage structure. The discharge of the pipe or closed channel under the filled condition computed with the friction slope equal to the constructed slope. For discharges greater than the capacity, the friction slope exceeds the constructed slope.

**C Coefficient -** Roughness coefficient used in the Hazen-Williams Equation.

**Channel Invert -** Lowest point on the surface of a channel cross section.

**Channel Slope -** Longitudinal slope in the channel. Also the vertical drop divided by the channel length. In irregular channels, the vertical drop is measured from low point to low point.

**Check Valve -** When this check box is toggled, flow can only travel from the From Node to the To Node in a pressure pipe.

**Click -** To quickly press and release one of the mouse buttons.

**Closed Channel -** A channel with a perimeter that forms a continuous closed boundary.

**Constructed Slope -** The average slope of the pipe inverts. The difference in the invert elevations between the upstream end and downstream end of the pipe divided by its length.

**Context Menu -** A pop-up menu opened by a **right** mouse click on a project element or data entry field. Commands on the context menu are specific to the current context state of the selected item.

**Contraction -** Adjustment coefficient used in the AASHTO equation to account for contraction of the flow at the entrance in the outlet pipe.

**Control Status -** A pressure pipe can be either Open or Closed. Open means that flow occurs in the pipe and Closed means that no flow occurs in the pipe.

**Conveyance Element -** A pipe or channel used to transport water.

**Cover -** Distance between the crown (soffit) of the pipe and the ground surface elevation.

**Critical Depth -** Depth of water in the channel for which the specific energy is at its minimum.

Rectangular:

$$Y_c = \left( \frac{Q^2}{gT^2} \right)^{1/3}$$

Non-rectangular (implicit solution, solving for $Y_c$):

$$Y_c = gA^3 - Q^2T$$

Where:

| | | |
|---|---|---|
| $A$ | = | Flow Area ($m^2$, $ft^2$) |
| $g$ | = | gravitational acceleration ($m/s^2$, $ft/s^2$) |
| $Q$ | = | discharge ($m^3/s$, cfs) |
| $T$ | = | Top Width (m, ft) |
| $Y_c$ | = | Critical Depth (m, ft) |

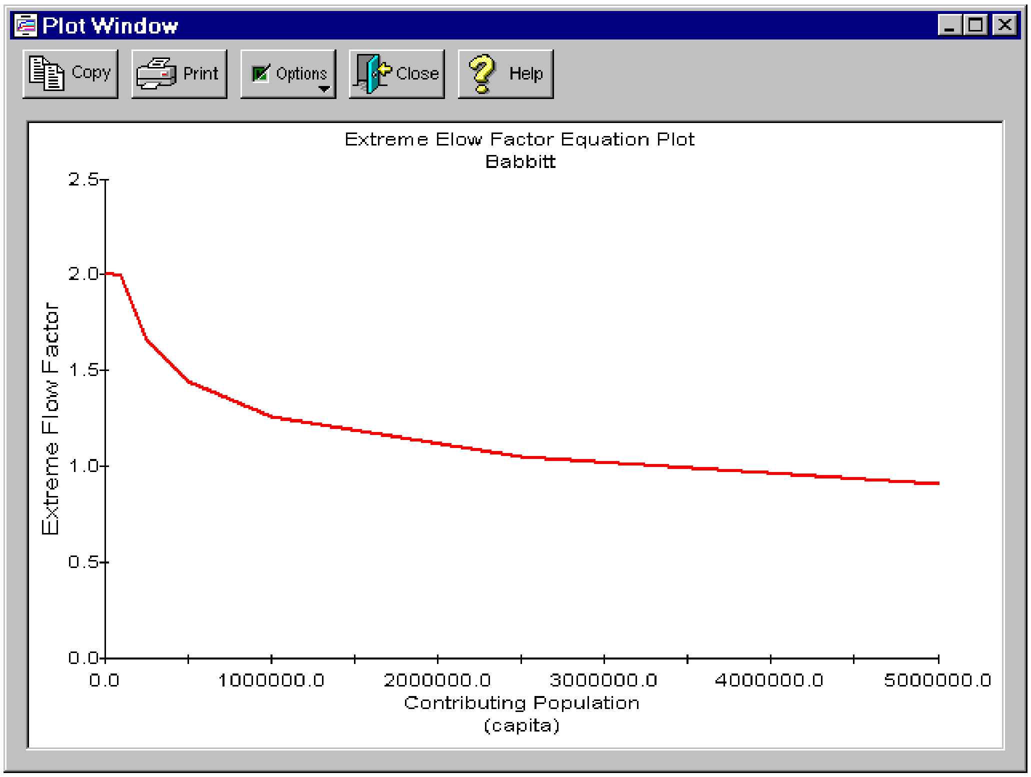
**Critical Flow -** Flow through a channel for which the specific energy is at its minimum.

**Critical Slope -** Channel or pipe slope for which the uniform flow is critical.

**Crosshair -** Shape of the cursor that looks like a plus sign ( + ).

**Crown (or Soffit) -** Top edge or the highest point of the pipe opening.

**Cutoff Value -** The cutoff value defines the maximum possible extreme flow factor for peaking methods or the minimum possible extreme flow factor for daily minimum methods. The generic equation used to calculate the extreme flow factor converges towards infinity as the population approaches 0. The cutoff value prevents unreasonably large extreme flow factors. The figure shows a cutoff value of 2.




**For minimum daily flows, extreme flow factor methods cutoff value limits how small an extreme flow factor can be.**

**Database Connections -** A connection represents a group of database links. There may be just a single linked external file within a connection, or there may be hundreds of external file links within a single connection.

**DBMS -** An acronym which stands for Database Management System. These systems can be relational (RDBMS) or non-relational.

**Depth -** See flow depth.

**Design Point -** The design point of a pump is the point at which the pump was originally intended to operate, and is typically the best efficiency point (BEP) of the pump. At discharges above or below this point, the pump is not operating under optimum conditions.

**Design Sump Depth -** A positive value will design a gravity structure with a sump elevation the Design Sump Depth below the downstream pipe invert elevation. In other words, the design sump depth is the distance below the lowest pipe invert that you wish to set the sump.

**Diameter -** Inside diameter of a circular channel, unless stated otherwise.

**Discharge -** Volumetric rate of flow given in units of length$^3$/Time.

**Display Precision -** In worksheets, rating tables, curves, and cross sections, the rounding of numbers and the number of digits displayed after the decimal point.

**Diversion -** A gravity node that diverts a portion of the flow out. A diversion element has two outlets while standard gravity elements have only one downstream pipe.

**Diversion Rating Curve Table -** A table that defines diverted flows as a function of total upstream flow.

**Diversion Target -** Destination for flows diverted out of a diversion element.

**Double-Click -** To click one of the mouse buttons (usually the left button) twice in rapid succession.

**Drag -** To hold down one of the mouse buttons (usually the left button) while you move the mouse.

**Duration -** The amount of time modeled during an Extended Period Simulation.

**Element -** An object, such as a manhole, outlet, or gravity pipe in a drawing.

**Elevation -** The elevation of an element is the distance from the datum plane to the center of the element. Elevations are often referenced with mean sea level as the datum elevation.

**Elevations Considered Equal Within -** The correction factor for relative flow (CQ) is applied only to situations where there are two or more pipes entering the structure at approximately the same elevation. Elevations Considered Equal Within is used to determine the maximum differences in pipe elevation for which the pipes are still considered approximately the same elevation.

**Energy -** Total energy of flow with reference to a datum. Computed for closed channels as the sum of channel centerline height above datum, piezometric height, and the velocity head. Computed for open channels as the sum of channel invert height above datum, the flow depth, and velocity head.

**Energy Equation -** The energy relationship between the downstream and upstream end of a pipe is:

$$\frac{V_1^2}{2g} + \frac{P_1}{?} + z_1 + h_G = \frac{V_2^2}{2g} + \frac{P_2}{?} + z_2 + h_L$$

Where:
- $V$ = fluid velocity (m²/s, ft²/s)
- $g$ = gravitational acceleration (m/s², ft/s²)
- $P$ = pressure (N/m², lb/ft²)
- $\gamma$ = specific weight of the fluid (N/m³,lb/ft³)
- $z$ = elevation at the centroid (m, ft)
- $h_G$ = head gain, such as from a pump (m, ft)
- $h_L$ = combined headloss (m, ft)

**Energy Grade Line -** Sum of datum (base elevation), velocity head, and pressure head at a section.

**Energy Slope -** The energy grade at each end of the pipe is computed by adding the velocity head component to the hydraulic grade. The energy slope is calculated by dividing the change in energy grade by the length of pipe.

**Exit Discharge -** Discharge in the pipe downstream of the structure.

**Exit Velocity -** Velocity at the upstream end of the pipe downstream of the structure.

**Exit Velocity Head -** Velocity head at the upstream end of the pipe downstream of the structure.

**Expansion -** Adjustment coefficient used in AASHTO equation to account for expansion of the flow on the exit from an incoming pipe.

**Extended Period Simulation (EPS) -** A calculation type where the model is analyzed over a specified duration of time. See also Steady-State Analysis.

**Extension -** The period and up to three characters at the end of a filename. An extension usually identifies the kind of information the file contains. For example, files you create in AutoCAD have the extension *.DWG.

**External Files -** Any file outside of the program that can be linked to. These include database files (such as FoxPro, Dbase, or Paradox), spreadsheets (such as Excel or Lotus), and ODBC connect strings. Throughout the documentation, all of these file types will be referred to as "databases" or "external files" interchangeably.

**Extrapolate -** Infer a value based on other values in an interval as in interpolation, with the value lying outside the known range of values.

$$\frac{Y_u - Y_1}{Y_2 - Y_1} = \frac{X_K - X_1}{X_2 - X_1}$$

Where: $Y_U$ = unknown value in Y

$X_K$ = known value in X

$X_1, X_2, Y_1, Y_2$= known table values

**Field Links -** Within each database table, the field links define the actual mapping between model element attributes and columns in the database.

**Flow** - Volumetric rate of flow given in units of length³ / time in the pipe or channel. Common units of flow are cfs, gpm, liters/sec, and cms.

**Flow Area -** Cross sectional area of flow.

**Flow Depth -** Distance from water level to low point of channel bottom.

**Flow Type -** The flow is defined as:

- Supercritical if $F > 1$
- Subcritical if $F < 1$
- Critical if $F = 1$

Where: F = Froude number.

**Friction Factor -** Friction Coefficient used in the Darcy-Weisbach Formula.

**Friction Slope -** Given a depth, roughness, section shape, friction method, and discharge, the friction slope is the computed slope that would be required to convey the specified discharge under uniform flow conditions. Under uniform flow conditions, depth and flow area are constant and the friction slope, the actual or constructed slope, and the energy slope are all equal.

Under gradually varied flow conditions, the depth of flow is changing along the channel and the friction slope is also varying. In the model's gradually varied flow solution, the friction slope between an upstream and downstream location along a prismatic channel is computed by taking the average of the uniform flow based friction slopes calculated at the beginning and ending stations of the sub-reach.

**Froude Number -** The Froude number is a unitless value representing the ratio of actual fluid velocity to wave celerity. It is computed as:

$$F = \frac{V}{\sqrt{g \cdot D}}$$

Where:
- $F$ = Froude number (unitless)
- $V$ = Velocity (m/s, ft/s)
- $D$ = Hydraulic depth (m, ft)
- $g$ = Gravitational acceleration ($m/s^2$, $ft/s^2$)

At critical depth, the Froude number is equal to 1.0.

**Full Flow -** Closed channel is flowing full, or entire channel perimeter is touched by flowing water (wetted).

**Full Flow Capacity -** The computed discharge when a closed channel is flowing full.

**Full Flow Slope -** The computed channel slope that would produce full flow.

**Global Diverted Flow -** The diverted flow entering a gravity node from one or more nodes located in other networks.

**Gravity -** Our products use these constants for all equations with g, gravity.

US: 32.174 $ft/s^2$

Metric: 9.81 $m/s^2$

**Ground Elevation -** The elevation of the ground surface at a node.

**Headloss -** Loss of energy due to friction and minor losses.

**Headloss Coefficient -** See Headloss Coefficient - Standard Method.

**Headloss Gradient -** Headloss in a pipe or channel represented as a slope, or gradient. This allows you to more accurately compare headlosses for pipes of different lengths.

**Headloss Method -** This program use one of the following methods to calculated headloss across a structure: Standard Headloss Method, Absolute Headloss Method, HEC-22 Energy Headloss Method or AASHTO Headloss Method.

**HEC-22 Benching Method -** Specifies which correction factor for benching is to be used, as specified in table 7-6 p. 7-19 of the FHWA HEC-22 manual used in the HEC-22 Energy Method.

**HEC-22 Energy Loss Method -** See Headloss HEC-22 Energy Method.

**HGL -** See hydraulic grade line.

**HGL In/Out**

- **HGL In:** The hydraulic grade at the downstream end of the incoming pipe section.
- **HGL Out:** The hydraulic grade at the upstream end of the outgoing pipe section.

**Hydraulic Grade -** See hydraulic grade line.

**Hydraulic Grade Line -** Hydraulic grade is computed using Gradually varied flow analysis in free surface conditions or by computing a pressure backwater starting from the submerged hydraulic grade at the downstream end of the pipe to the upstream end of the pipe, or a combination of free-surface and pressure. In open channels the hydraulic grade is equal to the water surface elevation.

**Hydraulic Radius -** Flow area divided by wetted perimeter.

**Hydraulic Time Step -** The time increment between hydraulic backwater analyses in the gravity portions of the system during an Extended Period Simulation.

**Hydrograph -** A graph of discharge vs. time.

**Hydrologic Time Step -** The time increment used when routing hydrographs in the gravity portion of the network, as well as the time increment used when running the pressure calculations.

**Inflow -** The Inflow is specified at a manhole or wet well as the total amount of wet weather inflow. Together with the infiltration along the pipes, the inflow forms the wet weather part of the sewer load.

**Interpolate -** A way of estimating a value between two known values assuming a linear relation.

$$\frac{Y_u - Y_1}{Y_2 - Y_1} = \frac{X_K - X_1}{X_2 - X_1}$$

Where: $Y_U$ = unknown value in Y

$X_K$ = known value in X

$X_1$, $X_2$, $Y_1$, and $Y_2$ = known table values

**Invert -** Bottom edge (lowest point) of the pipe opening. Sometimes referred to as the flow line.

**Invert Elevation -** The elevation at the bottom of the pipe. The invert elevation is the lowest point of the pipe opening.

**Junction -** Two or more pipes coming together.

**Kinematic Viscosity -** Viscosity divided by the mass density given in units of length$^3$/time, thus the term, kinematic.

**Known Flow -** Use the Known Flow field to model discharges at locations in the network where you have computed the hydrology using an external method (e.g. TR-55 tabular method).

During a Steady State analysis known flows are not additive except at network junctions. The Known Flow component will remain constant until it encounters a downstream Known Flow of a different value.

During an Extended Period Simulation known flows are treated as a constant inflow hydrograph and lumped in with other hydrographs entering at the same point. Known Flows are additive during Extended Period Simulations.

**Kutter's n Coefficient -** Roughness coefficient used in Kutter's Formula.

**Label -** A label is the reference by which an element will be referred to in reports, error messages, tables, etc. It is the unique "name" for your element.

**Length -** Distance from one end of the pipe to the other end of the pipe.

**Loading Pattern -** A series of multipliers over time which when applied to a single base load generate a hydrograph load.

**Local Diverted Flow -** Flow entering a gravity node that was diverted to that node from another node in the same network.

**Local Infiltration -** Infiltration entering the pipe as defined in the infiltration and rate sections.

**Local Total Infiltration -** Sum of the additional infiltration and local infiltration.

**Log Axis Scaling -** Compresses the values on the X and/or Y axis to the nearest power of 10. The numbers shown on the axes are common logarithms of the given variable.

**Manning's Coefficient -** Roughness coefficient used in Manning's formula.

**Material -** The material field is for selecting the pipe's construction material. This material will be used to determine a default value for the pipe's roughness.

**Matchline Offset -** Used to design invert elevations in and out of a gravity structure. The specified value will produce a corresponding drop between the upstream pipe invert elevations and the downstream pipe invert. A drop such as 0.1 ft is typically used to compensate for the junction headloss. This drop is applied either at the crown or at the invert of the pipes, depending on the pipe matching option you selected (crowns or inverts).

**Maximum Discharge -** The maximum theoretical discharge that could occur for a closed channel using a given hydraulic computation method. For closed circular channels, this discharge occurs at 0.938 x Diameter. Any increase in depth will decrease the discharge, which is why the full flow discharge is less than the maximum discharge for a circular channel. For a detailed explanation of this effect, see Ven Te Chow's *Open-Channel Hydraulics*.

**Maximum Extended Operating Point -** This point is the absolute maximum discharge at which the pump can operate, adding zero head to the system. This value may be computed by the program, or entered as a custom extended point.

**Maximum Operating Point -** The maximum operating point of a pump is the highest discharge for which the pump is actually intended to run. At discharges above this point, the pump may behave unpredictably, or the pump's performance may just decline rapidly.

**Minor Loss Coefficient -** Coefficient K used in the equation below, which is the equation most commonly used for determining the headloss in a fitting, valve, meter, or other localized component:

$$h_m = K\frac{V^2}{2g}$$

Where: $h_m$ = Loss due to the minor loss element (m, ft)
$V$ = Velocity (m/s, ft/s)
$g$ = Gravitational acceleration ($m/s^2$, $ft/s^2$)
$K$ = Loss coefficient for the specific fitting

**Mouse Buttons -** The left mouse button is the primary button for selecting or activating commands.

The right mouse button is used to activate (pop up) context menus and help.

**Mouse button functions can be re-defined using the Windows Control Panel.**

**Normal Depth -** For a prismatic channel section under a given constant discharge, the depth of flow that results for a specific channel slope. In subcritical flow conditions, the normal depth is greater than the critical depth. In supercritical flow conditions the normal depth is less than the critical depth.

**Number of Sections -** Number of parallel pipes of the same size and type.

**Object -** An icon on the tool palette that represents an element, such as a pipe, outlet, or junction in a drawing.

**ODBC -** ODBC (which stands for "Open Database Connectivity") is a standard programming interface developed by Microsoft for accessing data in relational and non-relational database management systems (DBMS's).

**OK Button -** When you click on this button the command chosen is carried out or modifications to data in dialog boxes are subsequently stored.

**On/Off Status -** The status of a pump can be either on or off. On means that flow will occur in the downstream direction, and the pump will add head to the system according to it's characteristic curve. Off means that no flow will occur, and no head will be added.

**Open Channel -** A channel with a free top surface.

**Open/Closed Status -** The status of a pipe can be either open or closed. Open means that flow can occur in either direction. Closed means that no flow will occur through the pipe.

**Opening Area -** Area of the orifice opening.

**Outflow -** Calculated flow being pump out of the wet well.

**Overflow Diversion Target -** Flow diverted out of an element can be transferred to another element in the system or lost from the system. Overflow indicates that the flow diverted out of the element is lost from the system.

**Patterns -** See Loading Patterns.

**Pattern Loads -** A type of load that consists of a single base load and a loading pattern that describes how that load varies over time.

**Percent Full -** Used in closed channels as a measure of flow depth divided by maximum depth.

**Piezometric Height -** Height that liquid rises to in a piezometric tube installed at the centerline of a closed channel.

**Pipe Crown Elevation -** Elevation of the crown of the pipe calculated as pipe invert elevation plus the height or diameter of the conduit. Also called crown (or soffit).

**Pipe Invert Elevation -** Elevation at the bottom of the pipe or channel.

**Point -** To move the mouse until the pointer on the screen is where you want it to rest.

**Population Equivalent -** Count of adjusted population for each loading unit of the load.

**Power -** A pump's power represents the water horsepower, or horsepower that is actually transferred from the pump into the water. Depending on the pump's efficiency, the actual power consumed (brake horsepower) may vary.

**Pressure -** Pressure measured at the specified elevation of an element.

**Pressure Head -** Energy due to the pressure of a liquid. For open channel flow, this value is zero.

$$\frac{p}{\gamma}$$

Where: $p$ = Pressure (Pa, psi)
$\gamma$ = Specific weight ($N/m^3$, $lb/ft^3$)

**Profile Description -** Specifies the type of flow profile in the pipe (such as S2 curve, M1 curve, etc.).

**Pull-down Menu -** A menu of available commands or actions you can carry out. A pull-down menu is usually selected from the menu bar at the top of the main program window.

**Pumped Flow -** Portion of the total flow that comes from a pump located in the upstream network.

**Pump Status -** A pump can have two different status conditions: On (normal operation), Off (no flow under any condition).

**RDBMS -** An acronym, which stands for Relational Database Management System.

**Relative Speed Factor -** The pump's relative speed factor defines the characteristics of the pump relative to the speed for which the pump curve was entered (in accordance with the affinity laws). A speed factor of 1.00 would indicate pump characteristics identical to those of the original pump curve.

**Reynold's Number -**

$$R_e = \frac{4VR}{\nu}$$

Where:
- $\nu$ = kinematic viscosity ($m^2/s$, $ft^2/s$)
- $R$ = hydraulic radius (m, ft)
- $R_e$ = Reynold's number
- $V$ = velocity (m/s, ft/s)

A high Reynold's number (larger than 4000) indicates turbulent flow, while a low one (between 0 and 2000) indicates laminar flow. The range 2000-4000 corresponds to a transitional regime.

**Rim Elevation -** The top elevation of a manhole structure. This elevation is typically flush with the ground surface. In some cases, the rim elevation may be slightly below the ground surface elevation (sunk) or slightly above the ground surface elevation (raised).

**Roughness -** The roughness coefficient based on the roughness method used.

**Roughness Coefficient -** Roughness value for the channel or pipe. With the exception of the Manning's Roughness Coefficients, the default values used are estimates. Most numbers that we show can be traced to literature; however, the values shown for methods other than Manning's are Haestad Method's estimates. Little information is available for these values and engineering judgment should prevail.

Algorithm by which a flow hydrograph is transformed to account for the transforming affects caused by the hydrograph traveling through a conduit or an open channel.

**Scientific Notation -** Numbers expressed as products consisting of a number between 1 and 10 multiplied by an appropriate power of 10.

**Section -** A cross section perpendicular to the flow through a channel or pipe.

**Section Shape -** The section type or geometric shape of the pipe. Haestad Methods supports circular pipes, box pipes, arch pipes, horizontal and vertical ellipses.

**Select -** To click the mouse button while pointing the cursor at an element in a list or at a command button.

**Service Area -** Total area contributing to sanitary loads at a node. The contributing area is specified through area-based unit sanitary load.

**Shaping Adjustment -** Adjustment coefficient used in the AASHTO equation for structure headloss calculation (refer to the equation in the Help) to account for partial diameter structure shaping (equivalent to Half and Full in HEC-22). If structure shaping is used then the headloss is decreased by this factor (50% default).

**Shortcut Keys -** Combination of keys allowing you to carry out menu commands instead of using the mouse.

**Shutoff Point -** The shutoff point is the point at which the pump will have zero discharge, and is typically the maximum head point on a pump curve.

**Size -** Inside diameter of a pipe section for a circular pipe. Dimensions of a box section or pipe arch (width x height).

**Slope -** Longitudinal slope in the channel. Non-uniform flow may have two types of slopes: friction slope and construction slope.

**Specific Energy -** Sum of the elevation head and velocity head as related to the section of a channel bed.

$$E = Y + \frac{V^2}{2g}$$

Where: $g$ = gravity ($m/s^2$, $ft/s^2$)
$V$ = velocity (m/s, ft/s)
$Y$ = depth (m, ft)

**Specific Weight -** The weight of a unit volume of a substance.

**Standard Headloss Method -** The Standard Method estimates headloss through junction chambers or manholes of a sanitary sewer system. See Headloss Coefficient - Standard Method.

**Station Number -** Station numbers are calculated from the network outlet, moving upstream along the pipe lengths. The Station Number format is specified by right clicking in the Station Number field and selecting Station Number Properties.

**Status Line -** The pane at the bottom of the window that shows information such as the coordinates for the current location of your cursor and the toolbar command being used.

**Steady State Analysis -** A calculation type where the model is analyzed for a single instant in time. See also Extended Period Simulations.

**Sump Elevation -** Elevation of the bottom of a manhole or outlet structure.

**Sub Menu -** A Sub Menu is a list of related options that is typically reached by selecting a pull-down menu item.

**Sump -** Elevation at the lowest point in a manhole or outlet structure.

**System Additional Infiltration -** Cumulative additional infiltration resulting from the upstream pipe network.

**System Infiltration -** Cumulative infiltration resulting from the upstream pipe network.

**System Total Infiltration -** Cumulative total infiltration resulting from the upstream pipe network.

**Table Links -** A table link must be created for every database table (or spreadsheet worksheet) that is to be linked to the model. Any number of Table Links may reference the same database file.

**Tailwater Elevation -** Water elevation downstream of the outlet.

**Task List -** A list of all the applications that you are currently running in Windows. The Task List lets you switch among applications, rearrange their windows, or quit them altogether.

**Tick Interval -** Distance between ticks on a graph.

**Top Width -** Length of the "free" top surface on the flowing cross section. For a cross section flowing full, this value is zero.

**Total Adjusted Population -** Non-population-based sanitary loads can be peaked by population-based extreme flow factor methods using Adjusted Population. Adjusted Population is specified based on Population Equivalent defined in a non-population-based unit sanitary load. It is a product of total non-population loading unit count and Population Equivalent.

**For example, 'Industry ABC' defines unit sanitary loads as a function of unit service area in hectares. For each unit service area Population Equivalent is 1000 capita. If a node has service area of, say, 25.3 hectares then the equivalent population used for peaking of this load is 25,300 capita.**

**Total Base Load -** Total base load is the sum of base loads of all sanitary loads in the sanitary load collection. Base load represent the average sewer sanitary load before it is transformed (peaked) using an extreme flow factor method.

**Total Diverted Flow -** The sum of the global and local diverted flow entering a gravity node.

**Total Flow -** Total Flow at the structure going through an element of the network.

**Total Population -** Population resulting from loads that are population based.

**Total Sanitary Flow -** Total Flow at a node resulting from sanitary sewage generated during dry weather from the network upstream of a given node.

**Total Wet Weather Flow -** Total Flow at a node resulting from the intrusion of rainfall water into the sewer system from the upstream network. Wet weather load consists of groundwater infiltration and rainfall inflow. Groundwater infiltration occurs in gravity pipes while inflows occur at manholes and wet wells.

**Uniform Flow -** Equilibrium flow for which the slope of total energy equals the channel slope.

**Unit Sanitary Load -** Loading unit count representing the local count of loading units for a specified unit sanitary load.

**Unit Load -** Sewage flow generated by one loading unit.

**Units -** Type of measurement displayed in worksheets, rating tables, curves, and cross sections.

**User Specified Tailwater Elevation -** A tailwater condition that typically occurs when the collection system discharges to a pond or other receiving body of water whose water surface elevation is greater than the pipe's crown elevation.  he tailwater elevation for the network can be specified directly in the Tailwater Elevation field. Although it is typically greater than the crown elevation, it can be any value.

**Velocity -** Linear measure of flow rate given in units of length/time.

**For weir and orifices, the velocity field is for the velocity of the water through the hydraulic structure.**

**Velocity Head -** Energy due to the velocity of a liquid:

$$\frac{V^2}{2g}$$

Where: $g$ = acceleration of gravity ($m/s^2$, $ft/s^2$)
$V$ = velocity (m/s, ft/s)

**Viscosity -** Property measuring the fluid resistance to shear.  Molasses and tar have relatively high viscosity and water and air have relatively low viscosity.

**Water Elevation -** Elevation of the water surface, also called the hydraulic grade in open channel flow.

**Water Surface Elevation -** Elevation of the channel's flowing surface, usually given in mean sea level (MSL).

**Weight -** The mass of a substance times the gravitational acceleration.

**Wetted Perimeter -** Perimeter of flow that travels against a solid boundary.  For a partially full pipe, the wetted perimeter includes all of the flow perimeter except for the top segment, which has a "free" surface.

**X Coordinate -** Coordinates are distances perpendicular to a set of reference axes.  Some areas may have predefined coordinate systems, while other coordinate systems may be arbitrary.

Coordinates may be presented as X and Y values, or may be defined as Northing and Easting values depending on individual preferences.

**Y Coordinate -** Coordinates are distances perpendicular to a set of reference axes. Some areas may have predefined coordinate systems, while other coordinate systems may be arbitrary.

Coordinates may be presented as X and Y values, or may be defined as Northing and Easting values depending on individual preferences.

# References

*AASHTO Model Drainage Manual*, American Association of State Highway and Transportation Officials, 1991.

American Society of Civil Engineers, *Gravity Sanitary Sewer Design and Construction*. American Society of Civil Engineers, New York, 1982.

Benedict, Robert P. *Fundamentals of Pipe Flow*. New York: John Wiley & Sons; 1980.

Brater, Ernest F.; King, Horace Williams. *Handbook of Hydraulics*. New York: McGraw-Hill Book Company; 1976.

Brown, S.A., S.M. Stein, and J.C. Warner, *Urban Drainage Design Manual. Hydraulic Engineering Circular No. 22*, U.S. Department of Transportation, Federal Highway Administration, Washington, D.C., 1996.

Chow, Ven Te. *Open-Channel Hydraulics*. New York: McGraw-Hill Book Company; 1959.

*Computer Applications in Hydraulic Engineering*. Third Edition. Haestad Press: Haestad Press; 1999.

*CulvertMaster User's Guide*. Connecticut: Haestad Methods; 1998.

D. Earl Jones, Jr. *Design and Construction of Sanitary and Storm Sewers*. ASCE Manual of Practice, No. 37; 1970.

*Essential Hydraulics and Hydrology*. Connecticut: Haestad Press; 1998.

Featherstone, R.E.; Nalluri, C. *Civil Engineering Hydraulics*. New York: Granada.

*FlowMaster PE User's Guide*. Connecticut: Haestad Methods; 1998.

French, Richard H. *Open-Channel Hydraulics*. New York: McGraw-Hill Book Company; 1985.

Practical Guide to Hydraulics and Hydrology. Connecticut: Haestad Press; 1997.

Hwang, Ned H. C.; Hita, Carlos E. *Hydraulic Engineering Systems*. New Jersey: Prentice-Hall, Inc.; 1987.

Hydraulic Research Station, *Velocity Equations for Hydraulic Design of Pipes*, Metric Edition, HMSO, London, 1951 (10/81).

Hwang, Ned H.C. and Hita, Carlos E., *Hydraulic Engineering Systems*, Prentice-Hall Inc, New Jersey, 1987.

Roberson, John A., John J. Cassidy, and Hanif M. Chaudhry, *Hydraulic Engineering*, Houghton Mifflin Company, Massachusetts, 1988.

Roberson, John A. and Clayton T. Crowe, *Engineering Fluid Mechanics (4th Edition)*, Houghton Mifflin Company, Massachusetts, 1990.

Simon, Andrew L., *Practical Hydraulics*, John Wiley & Sons, Inc, New York, 1976.

Singh, V.P., Hydrologic systems, Volume I.- Rainfall-runoff Modeling. Prentice Hall, New Jersey, 1988.

Streeter, Victor L. and Wylie, E. Benjamin, *Fluid Mechanics*, McGraw-Hill Book Company, New York, 1985.

Todini, E. and S. Pilati, "A Gradient Algorithm for the Analysis of Pipe Networks", *Computer Applications in Water Supply, Volume 1 - Systems Analysis and Simulation*, ed. By Bryan Coulbeck and Chun-Hou Orr, Research Studies Press LTD, Letchworth, Hertfordshire, England.

Tchobanoglous, George, Wastewater Engineering: Collection and Pumping of Wastewater, McGraw-Hill, Inc, 1981, Zipparro, Vincent J. and Hasen Hans, *Davis' Handbook of Applied Hydraulics*, McGraw-Hill Book Company, New York, 1993.

# Index

## A


AASHTO 99, 258
- Bend Angle and Loss Section 150
- Bend Loss 264
- Bend Loss Original Equation 264
- Calculation Options 150
- Coefficients 150
- Contraction Loss 263
- Correction For Non Piped Flow 265
- Correction for Shaping 265
- Expansion Loss 265
- Headloss - AASHTO Method 263
- Headloss Method 150
- Shaping Method 317

Abbreviated Labels 124
Absolute Headloss 317
Absolute Method 99, 258, 259
Absolute Roughness 317
Add 74
- Bends in AutoCAD 230
- Junction Chambers 74
- New Elements 74
- Outlets 74
- Pipe Run 222
- Pipes 74
- Pressure Junctions 74
- Pumps 74
- Wet Wells 74

Add New Load Dialog 111
Advanced Options 110
Adverse Slope 251
Aerial View 80
Affinity Laws 268
Allow
- Multiple Sections Section 105

Allow Multiple Sections 281
Alternatives 45-49, 127, 128, 133, 136, 144, 306
- Boundary Conditions 136
- Children 45
- Design Constraints 136
- Editor 129, 140
- Infiltration and Inflow Loading 134
- Inheritance 308
- Initial Settings 138
- Known Flow 135
- Manager 128
- Manager Dialog 21
- Operational 139
- Physical 130
- Sanitary Loading 133
- Structure Headlosses 135
- User Data 139

Analysis Results Report 24, 171
Analysis Toolbar 28
Annotation 24, 53, 55, 167, 168
- Comparison Wizard 184
- Multipliers 67
- Size 235

Annotation Properties 168
Annotation Tab 178
Annotation Wizard 168
Arch Section 192, 193
Area
- Enclosed by Polyline 77
- Flow 322

Area-Based Unit Sanitary Load 197
ASCII Loading Data Format 296
ASCII Loading File Example 301
Assumptions 260
Attribute
- Annotation 167, 168
- Inheritance 308

Auto Prompting 65
AutoCAD 183
- Command Line 16, 17
- Commands 230
- Drawing Synchronization 228
- DXF 223, 224
- Element Scale 229
- Entities 225, 226
- Exporting DXF File 223
- Import SewerCAD 232
- Importing DXF Files 224
- Importing SewerCAD DXF Files 224
- Proxies 233
- Rebuild Figure Labels 227
- Right Click Context Menu Option 65
- Undo/Redo 231

AutoCAD Mode 13, 225
- Graphical Layout 226
- Project Files 227
- Toolbars 226

AutoCAD Mode Window 13
AutoCAD Sessions 63
Autodesk 225

Autodesk Civil Design Export 221
Automatic Design 21, 43, 104, 147, 280, 285
  Hydrographs 285
  Pattern Loads 285
Automatic Element Labeling 223
Available in Materials 194
Average Velocity 149, 255, 256, 317
  Weigthed 255
Average Velocity Methods 255

## B

Background Drawing 223
Background DXF (AutoCAD) 223
Backwater Analysis 147, 254
Barrels 326
Base Load 317
Base Load Dialog 112
Base Scenarios 139, 140
Batch Run 139, 141
Benching Correction 262
Bend Angle 96, 150, 317
Bend Loss 150, 264
  AASHTO 264
  Coefficient 317
Bends
  Adding in AutoCAD 230
Bernoulli Equation 244, 321
Bibliography 333
Blocks (AutoCAD Mode) 217, 224
Bolted Cover 90, 317
Borders 185
Boundary Conditions 105
Boundary Conditions alternative 136
Box Section 192, 193
Button
  Cancel 317
  Help 25
  Mouse 326
  OK 326

## C

C Coefficient 249, 317
Calculate Network 21, 33, 147
Calculated Hydraulic Grade 317
Calculated Hydraulics Section 94
Calculation
  Problem Summary Report 173
  Results Status 31
Calculation Options 148, 150
  AASHTO 150
  HEC-22 149, 150
Calculation Options Generic Structure Loss 150
Calculation Type 147
Calculations 144, 148, 244
Cancel Button 317
Capacity 317
  Analysis 256, 257
  Full Flow 323
Capacity Analysis 147
Capital Cost Reports 164
Change
  Column Label 223
  Entities to Pipes 19, 232
Channel
  Closed 318
  Invert 318
  Open 326
  Slope 318
Check Valve 318
Chezy's Equation 246, 247
Child 308
  Scenario 139, 140, 307, 309, 310
Circular Section 192, 194
Classification
  of Profiles 251
  Slope 250, 251
Clipboard
  Copy Table 126
  Copy to 29
Closed Channel 318
Coefficient
  AASHTO 150
Coefficients 287
  Kutter's n 325
  Manning's 325
  Roughness 328
Colebrook-White
  Equation 248
  Typical Values 291
Color Coding 24, 56, 169, 170, 186, 235, 236
Column
  Allow Duplicate 120
  Change Units 236
  Heading 124
  Table Customization 124
  Table Setup 119
Command Line (AutoCAD Only) 16
Commands (AutoCAD Mode) 230
Compare Scenarios 184
Composite Capacity Profile 256
Composite Inflow Dialog 135
Composite Loading Dialog 133

Composite Minor Loss 97
Configuration 64
Conjugate Gradient Method 273
Connection 199, 200
  Database 203, 204
  Editing 203
  Hiding 207
  Management 201
  Missing 207
  Shapefile 210
  Sharing 207, 208
Connectivity Tolerance 218
Conservation
  of Mass & Energy 270
Constant Increment Hydrographs 298
Constant Increment Patterns Section 296
Constant Power Pump 93, 269
Constants
  Gravitational 323
Constraint Based Design 104
Constraints 21, 104, 286
  Warning Messages 286
Constructed Slope 318
Construction Costs 110
Construction Costs Table 110
Context Menu 318
Context Menu Option 65
Contraction 318
Contraction Loss
  AASHTO 263, 264
Control Condition 102
Control Preview 102
Control Status 318
Control Symbols
  Visibility 227
Control Type 102
Controls 102
Converting Native AutoCAD Entities 231
Convex Routing 151, 274, 275
Convex Routing Tab 151
Coordinate
  X 66, 331
  Y 66, 332
Copy to Clipboard 29
  Table 126
Copy/Paste 19
Correction for
  Benching 262
  Flow Depth 261
  Non Piped Flow - AASHTO 265
  Pipe Diameter 261
  Plunging Flow 261
  Relative Flow 261
  Shaping - AASHTO 265
Cost 159
  Alternative 139
  Analysis 159
  Manager 159
Cost Manager - Button Section 160
Cost Manager - Center Pane 160
Cost Manager - Left Pane 161
Cost Tab 109
Cost Warnings Report 165
Count-Based Unit Sanitary Load 197
Cover 318
Coveyance Element 318
Create
  New Elements 74
  New Project 26
Creating Scenarios 142
Critical
  Depth 318
  Depth and Slope 250
  Flow 250, 319
  Slope 251, 319
Crosshair 319
  Location 30
Crown 319
  Matching 282
Crown Elevation
  Pipe 327
CulvertMaster 316
Cursor 319
Cursor Location 30
Curve
  Pump 93, 94, 267, 268, 269
Custom AutoCAD Entities 225
Custom Extended 269
Custom Sort 123
Customization
  Drawing 227
Customize
  Database 201
  Drawing 67, 68, 69
  Labels 124
  Libraries 189
  Tables 124
Cut 19
Cutoff Value 319


## D


Darcy Weisbach
  Colebrook-White Equation 248
  Equation 247, 248

Roughness Values 291
Data
Entry 308, 309
Data Entry 33, 113
Data Organization 127
Data Structure 127
Database 199, 200
Export 201, 202, 203
Import 202, 203
Management Systems 319
ODBC 206, 207
Synchronization Options 204
Table Link Editor 204
Database Connection 199, 203, 204, 319
Editor 203
Example 208
Manager 201
ODBC 207
Standard 201, 202, 203
DBMS 319
Default 113
Design Constraints 104, 105, 157
Default Design Constraints 21, 44, 157
Default Kinematic Viscosity 288
Delete 19, 20
Elements 77, 229
Runs 223
Table 118
Delete Selection Set 78
Demand
Multipliers 154
Depth 319
Critical 250, 318
Flow 322
Design 43-45, 104-105, 157, 280-282
Considerations 287
Constraints 105
Default Constraints 157
Drop Structures 283
Extended 157
Limit Section Size 282
Menu 21
Part Full 281
Priorities 283, 285
Design Considerations 286
Design Point 93, 269, 320
Desired Sump Depth 320
Detailed Report 24, 171
Diameter 320
Correction 261
Direct Step Method 253
Direction of Flow
in Gravity and Pressure Systems 276
Discharge 320
Junction 324
Known 324
Maximum 325
Outlet 90
Discharge-Based Unit Sanitary Load 197
Display Precision 70, 320
Display Tips
Change Units in a Column 236
Color Code Elements 235
Control Element/Label Sizing 235
Reuse Deleted Element Labels 235
Diversion 320
Diversion Network 183
Diversion Network Background Color 184
Diversion Network Options 184
Diversion Network Window 183
Diversion Parameters 101
Diversion Rating Curve Table 101, 320
Diversion Tab 100
Diversion Target 320
Double-click 320
Drafting 77, 185
Drag 320
Drawing
Options 63, 64, 67
Pane 15
Preview 220
Review 80, 81, 217
Scale 67
Setup (AutoCAD Mode) 227
Synchronization (AutoCAD Mode) 228
Drawing Tab 178
Drop Structures 283
Sanitary (Dry Weather) Loading 239, 241
Duration 320
DWG 63, 227
DXF 68, 223
Background 30, 68
Exporting from SewerCAD 224
Import 223
Import Techniques 224
Units 68
Dynamic Inheritance 307, 308
Dynamic Parameters 178


## E


Edit
Controls 102
Gravity Pipes 88
Junction Chambers 86, 87

- Manholes 86
- Outlets 88
- Pipe Run 222
- Pressure Junctions 88
- Pumps 87
- Wet Wells 87

Edit Elements 229
Edit Menu 19
Editable Table Columns 121, 122
Editing Elements 76
EGL 245
Element 85, 229, 320
- Annotation 167
- Delete Bend (AutoCAD Mode) 19
- Deleting 77, 229
- Editing 19, 86, 229
- Find 78, 79
- Insert Bend (AutoCAD Mode) 19
- Labeling 83, 223, 235
- Modify 229
- Morphing 74
- Moving 77, 230
- Numbering 82, 235
- Rotate (AutoCAD Mode) 19
- Scale (AutoCAD Mode) 19, 229
- Search 78
- Selection 75, 76, 77, 83
- Type 74, 75

Element Detailed Cost Report 164
Element Graphing Report 173
Element Properties 228
Elevation 103, 320
- Ground 323
- Invert 324
- Mode 66
- Pipe Crown 327
- Pipe Invert 327
- Rim 328
- Water 331
- Water Surface 331

Elliptical Section 192, 194
Email Address 11
Enclosed Area 77
Energy 320
- Balance 252
- Conservation 270
- Discontinuity 286
- Equation 245
- Grade Line 245
- Loss 149, 150
- Principle 244
- Specific 329

Energy Discontinuity 286
Energy Grade Line 321
Energy Slope 321
Engineering Library 189, 190, 191
- Available Materials 194
- Editor 190
- Extreme Flow Factor Method 191
- Manager 189
- Material 195
- Minor Loss 195, 196
- Section Size 192

Engineer's Reference 287
Enter Key Behavior 64, 65
Entities
- Change to Pipe 232

Entities in AutoCAD 225
Entity Conversion 232
Equation
- Bernoulli 321
- Chezy 246
- Kutter's 246

Equations 287
Error Messages
- Results Report 145

ESRI 199, 200, 215
- Export Shapefile 213
- Import Shapefile 211
- Shapefile Connection 210

Example Projects
- Tutorials 10

Excess Capacity Profile 256, 257, 258
Exit Discharge 321
Exit SewerCAD 19
Exit Velocity 321
Exit Velocity Head 321
Expansion 321
Expansion Loss
- AASHTO 265

Explode Elements (AutoCAD Mode) 230
Export 18
- Database 18, 19, 201, 202, 203
- DXF 17, 223
- Land Development Desktop 18
- Profiles in AutoCAD 183
- Runs 222
- Shapefile 18, 19, 213, 214
- Shapefile Link Editor 214
- Structure Mappings 223
- Table to ASCII 126
- to Autodesk Civil Design 221

Extended Period Section 148
Extended Period Simulation EPS 321
Extended Period Simulations Overview 274
Extension 321

External Files 321
Extrapolate 322
Extreme Flow Factor 33, 241, 242
  Equation Properties 191
  Method 191
  Table Properties 192
Extreme Flow Setup 156
  Manager 156

## F

Factor
  Extreme Flow 241, 242
  Friction 322
  Relative Speed 94
  Variable Peaking 242
Fax Number 11
Field
  Links 205
  Share 116, 117
Field Links 322
File
  Export Settings 222
File Management 63
File Menu 17
Files
  AutoCAD 227
Filter Tables 123
Find Element 19, 78, 79
Fitting Loss Coefficients 266
Fittings Library 195
FlexTables 40, 41, 117
FlexUnits 69, 70
  Table 71
Flooding 254
Flow 322
  Area 322
  Critical 250
  Depth 322
  Depth Correction 261
  Full 323
  Known 324, 325
  Pressure 249
  Regime 249
  Subcritical 250
  Supercritical 250
  Surcharged 249
  Type 322
  Uniform 331
Flow Diverted In 101
Flow Diverted Out 101
Flow Profile Methods 149
FlowMaster 316
Format 70
  User Data 114
Formulas 287
Free Outfall 254
Friction Factor 322
Friction Loss Methods 245
Friction Method Theory 66
Friction Slope 322
Frontwater Analysis 254, 255
Froude Number 323
Full Capacity Profile 256
Full Flow 323
  Capacity 323
  Slope 323
  Velocity 255

## G

General Information 89, 91
General Section 89, 91
General Status Information 30
Global Diverted Flow 323
Global Edit 122
Global Options 64, 65
Globe Button 6
Glossary 10
Go Button 147
Grade
  Hydraulic 324
Grade Line
  Energy 321
  Energy 245
  Hydraulic 324
  Hydraulic 245
Gradient Algorithm 270
  Derivation 271
Gradually Varied Flow 250
  Direct Step Method 253
  Standard Step Method 253
Graph Options 176
Graph Setup 173, 174
Graphic Annotation 22, 185
Graphical Editor 73
Graphical Layout
  AutoCAD 226
  Stand-Alone 73
Gravity 323
Gravity Hydraulics Tab 148
Gravity Pipe 88, 89
  Add 74
  Default Design Constraints Section 157

Infiltration/Inflow Loading 134
Gravity Pressure Interface Options 152
Gravity Structure
Design Constraints 137, 138
Gravity Structure Tab 158
Gravity to Pressure Network Transition 277
Ground Elevation 323
Grouping Elements 75
Selection Sets 77

## H

Haestad Methods 5, 11
Haestad Methods Knowledge Base 237
Haestad Methods Workshops 10
Hastad Methods Software 315
CulvertMaster 316
FlowMaster 316
PondPack 316
SewerCAD 315
StormCAD 316
WaterCAD 315
Hazen Williams Equation 249
Coefficients 293
Roughness Values 292
Head
Pressure 327
Velocity 331
Headloss 149, 150, 259, 323
Coefficient 195
Headloss - Generic Method 259
Headloss Coefficient 323
Manholes and Junctions 289
Headloss Gradient 323
Headloss Method 99, 149, 150, 258, 323
AASHTO 100, 258, 263
Absolute 99, 258, 259
HEC-22 Energy 99, 259, 260
Standard 100, 258, 259
HEC-22 99, 149, 258, 259, 260
Benching Method 323
Energy Loss Method 323
Height
Piezometric 327
Help 9, 10, 25, 26, 33, 235
How Do I 9
Technical Support 11
Tutorials 10
Help Button 25
Help Menu 25
HGL 245, 323
Above Ground / Rim 254
Structure 324
HGL Mode 66
Horizontal Section Size Properties 194
Horizontal Slope 251
Horse Power (Pump) 93
How Do I 9, 25, 235, 236
Profile Plot 236
Share Database Connections 207
How to Use Help 25
Hydraulic Grade Line 245, 323, 324
Hydraulic Jump 103
Hydraulic Radius 324
Hydraulic Results Section 98
Hydraulic Time Step 276, 324
Hydraulic Transition
from Gravity to Pressure Network 277
from Pressure to Gravity Network 278
Hydraulics Options 149
Hydrograph 324
Hydrograph Dialog 112
Hydrographs 173, 240
Hydrologic Time Step 276, 324
Hydrologic vs. Hydraulic Time Steps 276

## I

Identifying Gravity Pipes and Force Mains 276
Import 18
Database 201
DXF 18
DXF Files into AutoCAD 224
Land Development Desktop 17
Polyline to Pipe 18
SewerCAD 18, 19, 232
Shapefile Link Editor 212
Import Loading Data Dialog 295
Importing Loading Data Overview 295
Include In Cost Calculation 109
Infiltration 243
Additional 108
Direct 108
Local 109
System 109
Infiltration and Inflow Loading 108, 109, 243
Infiltration and Inflow Loading Alternative 134
Infiltration Hydrograph 108
Infiltration Pattern Load 108
Infiltration Tab 108
Infiltration Type and Rate Sections 108
Inflow 243, 324
Inflow Loading
Structure 135

Inflow Section 107
Inheritance 307, 308
  Dynamic 308
  Overriding 307
Initial Headloss Coefficient 260
Initial Settings
  Pressure Pipe 138
  Pump 138
  Wet Well 139
Initial Settings Alternative 138
Initialize
  Run List 223
Input 76
  Quick View 84
Input Data 85
Input Modes 66
Insert
  Elements 75
  Nodes 75
Installation Guide for Network License Versions 7
Installation Problems 4
Installing Haestad Methods Products 4
Internet Address 11
Interpolate 324
Interval
  Tick 330
Inventory 50, 172
Invert 324
  Channel 318
  Matching 282
Invert Elevation 96, 324
  Pipe 327
Inverts and Sizes 280


## J


Junction 289, 324
  Flooding 254
  Headloss Coefficients 289
Junction Chamber 23
  Add 74
Junctions
  Headloss Coefficients 289


## K


K Coefficients 195, 266
Kinematic Viscosity 324
Known Discharge 324
Known Flow 324
Known Loading 239, 243
Known Loading Alternative 135
Kutter Formula 290
Kutter's Equation 246
  n Coefficient 325
  Roughness Values 290


## L


Label 325
  Abbreviate 124
  Automatic 223
  Elements 83, 235
  Rotate 230
  Sizing 235
Label Rebuilding (AutoCAD Mode) 227
Labels Tab 178
Land Development Desktop
  Export Wizard 222
  Import Settings 221
  Import Wizard 220
  Unit System 222
Laws
  Affinity 268
  Conservation of Mass & Energy 270
Layer 220, 229
Layers Tab 178
Layout 33-35
  AutoCAD 226
  Network 73
Layout Options Tab 178
Layout Pipe Using Entity 232
Left click 328
Legend 24, 185, 186
  Quick View 84
Length 325
Lesson 5 Running an Extended Period Analysis 56
Lessons 33, 43, 55
  Automatic Design 43
  Creating a Schematic Network 33
  Presentation of Results 50
  Scenario Management 45
Level Mode 66
Library
  Editor 190
  Manager 190
  Material 195
  Minor Loss 195
Library System
  Extreme Flow Factor Method 191
  Section Size 193

Licenses 5
Limit Section Size 105, 282
Line 185
Enclosed Area 77
Linear System Equation Solver 273
Linear Theory Method 270
Link Color Coding 170, 235
List
Task 330
Loading 37-39, 41, 196, 197, 239, 243
Dry Weather 239
Known 239, 240, 243, 244
Wet Weather 239, 240, 242, 243
Loading Pattern 325
Local
Addional Infiltration 317
Infiltration 325
Units 125
Local Diverted Flow 325
Local Infiltration 109
Log Axis Scaling 325
Loss and Bend Angle Section 150
Losses
Minor 266

## M

Mailing Address 11
Manager
Extreme Flow Setup 156
Manholes 22, 86
Add 74
Mannings Equation 246
Manning's Coefficient 325
Roughness Values 290
Typical Values 293
Mass Conservation 270
Match Pipes 282
Matchline Offset 282, 325
Material 195, 325
Library 195
Maximum Discharge 325
Maximum Extended Operating Point 93, 325
Maximum Operating Point 93, 326
Menu 16, 17, 19, 20, 22, 24, 25
Context 318
Design 21
Edit 20
File 17, 18
Help 25
Options 113
Report 25
Shortcut Keys 16
Toolbars 16
Tools 22, 24
Messages 111
Metric 64, 70
Microstation 223
Mild Slope 251
Minimum
Allowed Value 70
Minimum Structure Headloss 149
Minimum System Requirements 3
Minor Losses 266
Coefficient 326
Fitting 267
Properties 195
Mix Units in a Tabular Report 125
Mixed Flow Profiles 253
Mode
Input 66
Scaled 67
Schematic 67
Stand-Alone/AutoCAD 13
Model 85
Morphing Elements 74
Move
Elements 76, 77, 230
Labels 231
Multiple
Pump Curve 269
Units 125, 126
Multiple Sessions 63
Multipliers 67

## N

Native AutoCAD Entities Converting 217, 231
Network Deployment Folder 9
Network Licensing 6
New 17, 18
Project 26
New Elements 74
Node
Color Coding 170, 235
Non-Construction Costs 110
Non-Population Based Unit Sanitary Load 196
Normal Depth 249, 326
Northing Easting Mode 66
Notation
Scientific 328
Scientific 70
Number
Froude 323

of Digits After Decimal Point 69
Reynold's 328
Station 329
Number of Sections 326

## O

Object 326
ODBC 206, 326
Offset Matching 282
OK Button 326
On-line Help 9, 25
Open Channel 326
Open Database Connectivity 206, 326
Open Project 17
Open/Closed Status 326
Opening Area 326
Operating Point Section 94
Operating Range 106
Operational Alternative 139
Options 30
AASHTO 150
Calculation 148
Drawing 67
Global 64
Graph 174, 175, 176
HEC-22 149
Pressure 151, 152
Profiling 181
Project 63, 66
Options Menu 113
Organize Data 127
Other Factors 150
Outflow 327
Outlet 23, 24
Control 254
Outlets 88
Add 74
Output 167
QuickView 84
Tables 117
Overflow Diversion Target(Overflow) 327
Overview 63

## P

Part Full Design 281
Pattern 152, 153
Demand
Multipliers.................................................. 154
Pattern ........................................................ 154
Extended Period Analysis 154
Pattern Editor 154
Pattern Graph 155
Time Steps 154
Pattern Load Dialog 112
Pattern Loads 240, 327
Pattern Manager 153
Pattern Setup 155, 156
Pattern Setup Manager 155
Patterns 327
Peaking Factor 156, 242
Percent Full 327
Perimeter
Wetted 331
Phone Number 11
Physical Properties 130
Junction Chamber 132
Manhole 131, 132
Pressure Junction 132
Pressure Pipe 130
Pump 132
Wet Well 132
Pie Charts 176
Piezometric Height 327
Pipe
Average Velocity 255
Bends 150
Controls 101, 102
Crown Elevation 327
Default Design Constraints 157
Design Constraints 104
Diameter Correction 261
Elevations 103
Exit Properties 100
Extended Design Section 157
Fittings 195
General Characteristics 95
Hydraulic Summary Section 96
Infiltration 243
Invert Elevation 96, 327
Length Rounding 66
Matching 104, 282
Profile 103
Roughness 96
Sizing 95
Splitting 75
Tool 74
User Defined 96
Pipe Costs Report 165
Pipe Layout Using Entity 232
Pipe Run
Add 222
Pipe Tool 22

Pipes 88
Add 74
Plan View 25
Plan View Reports 172
Plot Window 174
Plunging Flow Correction 261
Point 327
Polygon Enclosed Area 77
Polyline
Conversion Problem 220
Enclosed Area 77
Polyline to Pipe Conversion 217
Wizard 218
PondPack 316
Population Based Unit Dry Weather Load Properties 196
Population Equivalent 327
Power 93, 327
Precision
Display 320
Precision Display 70, 71
Predefined Reports 170, 171
Preferences 64
Presentation 167
Preserve Pumped Flows 152
Pressure 327
Head 244, 245
Pressure Flow 249
Free Surface Flow
and Transitional Flow.................................. 260
Pressure Head 327
Pressure Hydraulics Tab 151
Pressure Junctions 24, 88
Add 74
Pressure Junctions General Section 95
Pressure Mode 66
Pressure Options 151
Pressure Pipe 89
Add 74
Initial Settings 138
Pressure to Gravity Network Transition 278
Preview 186
Print 17, 19, 30
Preview 26, 29
Preview Window 186
Setup 17
Table 126
Problems with Setup or Uninstall 4
Profile 103, 104, 176, 177, 181, 182, 236
Axis 182
Classification 251, 252
Drawing Tab 182
Elements Tab 181
Excess Capacity 256, 257, 258
Full Capacity 256
Options Dialog 181
Plot 180, 236
Templates Dialog 177
Window 180, 181
Wizard 179, 180
Profile Background Tab 182
Profile Description 327
Profile Layers 182
Profile Methods Section 149
Profiles Manager 176, 177
Profiling 24, 182
Project
Inventory 50
Inventory Report 172
New 26, 27
Open 26, 27
Options 63, 66
Save 26
Setup Wizard 63
Summary 63, 64
Title 64
Project Detailed Cost Report 165
Project Element Summary Cost Report 165
Project Files 63, 227
Project Summary 17
Project Summary Cost Report 165
Prototype 63
Prototypes 113
Proxies 233
Publications 315
Pull-down Menu 16
Pulldown Menus 17
Pump 22, 87, 88
Add 74
Affinity Laws 267
Constant Horsepower 269
Curve 93, 267, 268, 269
Definition 93
Initial Settings 138
Initital Condition 94
Operating Point 267, 268, 269
Power 327
Power 93
Pumped Flow 327
Status 328
Theory 267
Type 269
Variable Speed 268
Pump Curve 93, 94, 267, 268

## Q


Quick Attribute Selector 71
Quick View Window 21


## R


Radius
  Hydraulic 324
Rainfall 243
Rapidly Varied Flow 253
RDBMS 328
Rebuild Figure Labels 227
Recent Files 17
Redefine SewerCAD Blocks 224
Redo 19, 30, 231
Reference
  Engineer's 287
References 333
Registering Network Programs 6
Registration 5, 25
Relabel Elements 81
Relabel Operations 82
Relational Database Management System 328
Relative Flow Correction 261
Relative Speed Factor 94, 328
Release Notes 25
Remove
  Columns 120
  Elements 77
  Haestad Methods Products 4
Removing Color Coding from labels imported from Pre-v3.5 files? 236
Report Menu 24
Reports 51, 52, 167
  Analysis Results 171
  Detailed 171
  Plan View 172, 173
  Predefined 170, 171
  Project Inventory 24, 172
  Scenario 25, 172
  Tabular 24, 172
Requesting Permanent Network License 7
Results 167
Review Drawing 80
Reynold's Number 328
Right Click Context Menu Option 65
Right Mouse Button 65
Rim Elevation 328
Rotate Labels (AutoCAD Mode) 230
Roughness 328
  Absolute 317
  Chezy's Equation 246
  Coefficient 195, 290
  Colebrook-White Equation 248
  Darcy-Weisbach Equation 247
  Hazen-Williams Equation 249
  Manning's Equation 246
Roughness Coefficient 328
Roughness Values 287
  Colebrook-White 291
  Darcy-Weisbach 291, 292
  Hazen-Williams 292, 293
  Kutter Formula 290
  Manning's 290
  Typical 293
Rounding 70
  Pipe Length 66
Routing Overview 274
Running the Model 147
Runs
  Delete 223
  to Export to Land Development Desktop 222
  to Import 221


## S


Sales 11
Sample Projects 10
Sanitary Dry Weather Flow Section 107
Sanitary Hydrograph Loads Section 299
Sanitary Loading Alternative 133
Sanitary Pattern Loads Section 297
Sanitary Unit Loads Section 299
Save 18, 19
  as Drawing*.dwg 228
  Project 26
Save As 17
Scale 67
  Elements (AutoCAD Mode) 229
Scaled Mode 67
Scaling
  Log Axis 325
Scenario 139, 141, 142, 184, 306, 307
  Alternatives 144
  Base 139, 140
  Batch Run 141, 142
  Child 307
  Comparison 184
  Editor 140, 144
  Inheritance 307, 308, 309
  Management 127
  Results 145
  Selection 140

Summary Report 172
Scenario
Management 21, 45-49, 141, 303, 313
Example 310
Scenario Wizard 142
Scenario Wizard - Step 3 143
Scenarios 45, 46, 48, 49, 50
Alternatives 45
Analysis Toolbar 28
Calculation 144, 145
Children 45
Comparison 45, 49
Management 45
Schematic 33, 34
Schematic Mode 67
Scientific Notation 70, 71, 328
Sealing Conditions 253
Search for Elements 78
Section 328
Materials 194, 195
Section Size 193, 194
Arch 193
Box 193
Circular 194
Horizontal Ellipse 194
Properties 192, 193
Vertical Ellipse 194
Section Type 328
Sections
Number of 326
Select 23, 328
By Selection Set 19, 77
Element Types 211
Elements 20, 75, 76
Field Links 205
Layer 229
Runs to Import 221
Text Style 229
Selecting Upstream / Downstream Elements 76
Selection Set 77, 78
Manager 21, 78
Selection Tool 22
Service Area 328
Set Field Options 69
Settings 64
Setup 4, 64, 227
Drawing Options 67
Global Options 64
Problems 4
Project Options 66
Prototypes 113
SewerCAD 1, 25, 26, 315
AutoCAD Custom AutoCAD Entities 225
Documentation 9
Elements 85
Exploring 13
How to Use it 1
Import 232
Menus 318
Toolbars 26
User's Guide 9
SewerCAD in AutoCAD 225
Files 227
SewerCAD Theory 239
SewerCAD VCR Controls 28
SewerCAD Window 13
Shapefile 208, 209, 210, 211, 212
Format 215
Shapefile Connection 208, 209, 210, 211, 212
Editor 209
Example 215
Export Example 214
Export Wizard 213
Import Example 212
Import Wizard 211
Link Wizard 210
Manager 208
Synchronization Options 212
Wizard 209
Shaping Adjustment 329
Shaping Method 317
Share Fields 116
Sharing Shapefile Connections between Projects 215
Sharing Templates Between Projects 179
Shortcut Keys 16, 329
Shutoff Point 93, 329
Simple Average Velocity 255
Simultaneous Path Adjustment Method 270
Size 329
Size Elements (AutoCad) 229
Sizing Pipes 280
Slope 329
Adverse 251
Channel 318
Classification 250
Constructed 318
Critical 250, 251
Friction 322
Full Flow 323
Horizontal 251
Mild 251
Steep 251
Snap Menu (AutoCAD Mode) 231
Soffit 319
Software Registration 5

Sort
- Custom 123

Sort By Network 123
Sorting Tables 122
Sparse Matrix 271, 273, 274
Specific Energy 329
Specific Weight 329
Splitting Pipes 75
Stand-Alone Mode and AutoCAD Mode 13
Standard Database Import/Export 201
Standard Extended 269
Standard Headloss Method 329
Standard Method 99, 100, 258, 259
Standard Step Method 253
Station Number 89, 91, 329
Status 30, 31, 62
- Bar 15, 30, 31
- Line 329
- Log 187
- Pane 64

Steady State Analysis 329
Steady State EPS Load Method Section 151
Steady State Section 147
Steep Slope 251
Sticky Tools 65
StormCAD 316
Stretch 77
Structure
- Flooding 254
- Headlosses 258, 259
- HGL 324
- Sump Elevations 283

Structure Energy Grade 286
Structure Headloss Alternative 135
Structure Mapping
- for Civil Design Import 221
- for Export 223

Sub Menu 329
Subcritical Flow 250
Suggestions 12
Sump 104, 329
- Elevation 283

Supercritical Flow 250
Support 11
Surcharged Flow 249
Surcharging 253
SWR File 63, 227
Symbol
- Size Multiplier 67

Symbol Visibility 69
Symbol Visibility (AutoCAD Mode) 227
Synchronize 209
- AutoCAD Mode 228
- Database Links 201
- Error 63
- Options 204
- Via ODBC 207

Synchronized Units 125
System
- International 64, 65, 70

System Additional Infiltration 330
System Cost Adjustments Table 161
System Infiltration 109, 330
System Requirements 3, 4
System Total Infiltration 330


# T


Table 117, 120, 121
- Change Units 125
- Copy to Clipboard 126
- Customization 124
- Editing 118
- Export to ASCII File 126
- Filtering 123, 124
- Flex Units 71
- Links 330
- Manager 117
- Mixing Units 125
- Navigation 121, 122
- Print 126
- Print Preview 126
- Properties 119
- Setup 119
- Two Rows 118
- Type 119

Tabular
- Report 117

Tailwater
- Elevation 330, 331
- Hydraulics 90
- User Specified 331

Task List 330
Technical Support 11
Templates 177-179
- Annotation Tab 178
- Drawing Tab 178
- Labels Tab 178
- Layers Tab 178
- Layout Options Tab 177
- Sharing Templates Between Projects 179

Templates Dialog 177
Text 231
- Style 229

Text Height 68

How Do I 235
Multiplier 67, 68
Theory 239, 244
Tick Interval 330
Time
Travel 256
Time-Based Loading 240
Times Section 300
T-intersections 219
Title
Project 63, 64
Tolerance 81
Tool Pane 64
Tool Pane Toolbars 26
Toolbar 16, 226
and Shortcut Keys 16
Toolbars
Buttons 26
Tools Menu 22
Top Width 330
Total
Adjusted Population 330
Base Load 330
Sanitary Flow 330
Flow 330
Population 330
Wet Weather Flow 330
Total Diverted Flow 330
Transition
from Gravity to Pressure Network 277
from Pressure to Gravity Network 278
Travel Time Computation 149, 256
Tutorials 10, 25, 26, 33
Two Row Table 118
Type Coercion 199
Types of Calculation 147

## U

Undo 19, 30
Undo/Redo Operations in AutoCAD 231
Uniform Flow 249, 331
Velocity 255
Uninstall 4
Problems 4
Unit Conversion 199
Unit Cost
Data Table 162
Function Formula 162
Function Manager 161
Table Coefficients 163
Unit Sanitary Loads 196, 197, 241, 331
Unit Load 331
Unit System 64, 65, 71
Units 64, 65, 68, 70, 71, 236, 331
Change in a Column 236
DXF 68
Local 125
Synchronized 125
Updates 5, 26
Upgrades 5
US Customary 65
US Customary Units 64, 70
Usage 191
User Data 113
User Data Alternative 139
User Defined Bend Angle Section 96
User Memos 111

## V

Variable Increment Hydrographs Section 298
Variable Increment Patterns Section 296
Variable Peaking Factor 242
Variable Speed Pumps 268
Velocity 331
Full Flow 255
Head 245
Simple Average 255
Uniform Flow 255
Weighted Average 255
Velocity Head 331
View
Menu 21, 22
Tabular 117
Viscosity 331
Kinematic 324
Visibility of Symbols 227

## W

Warning Messages 286
Water Elevation 331
Water Surface Elevation 331
WaterCAD 315
Weight 331
Specific 329
Weighted Average Velocity 255
Weighted Translation Routing 275
Welcome Dialog 25, 64
Wet Hydrograph Loads Section 299
Wet Pattern Loads Section 297
Wet Weather Loading 108, 239, 242

Wet Well 24
Initial Settings 138, 139
Wet Well Increment 152
Wet Wells 87
Add 74
Wetted Perimeter 331
What is New in SewerCAD? 2
White Table Columns 121
Width
Top 330
Window Color 65
Wizards 168, 213
Annotation Comparison 184
Polyline to Pipe 217, 218
Profiling 180
Project Setup 63, 64
Shapefile Connection 209
Shapefile Export 214
Shapefile Link 210
WYSIWYG 50

## X

X - Y Mode 66
X Coordinate 66, 89, 91, 331

## Y

Y Coordinate 66, 89, 91, 332

## Z

Zone Classification 251
Zoom 21, 22, 26, 27, 79