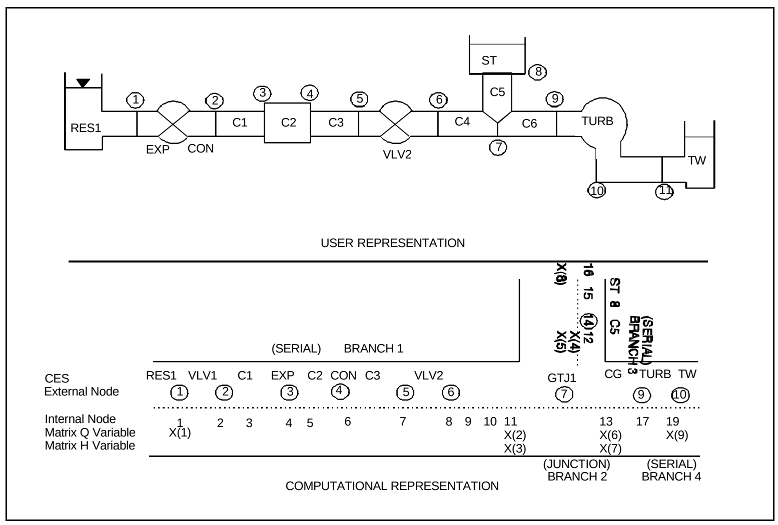
US Army Corps
of Engineers
Construction Engineering
Research Laboratories

USACERL ADP Report 98/129
September 1998

Water Hammer and Mass Oscillation (WHAMO) 3.0 User's Manual

Robert Fitzgerald and Vicki L. Van Blaricum

Fluid distribution systems and hydropower plants can be severely damaged by water hammer, a forceful slam, bang, or shudder caused by a sudden change in fluid pressure within the system. Water hammer can be mitigated by designing and operating these systems to minimize sudden changes in water velocity, which lie at the root of water hammer problems. WHAMO 3.0, a modeling and simulation application for desktop computers, was specifically developed for this purpose. WHAMO provides dynamic simulation of fluid distribution systems and hydropower plants. It calculates the time-varying flow and head in networks comprised of pipes, valves, pumps, tanks, and turbomachines. WHAMO Version 3.0 is a major upgrade that implements a graphical user interface for operation under Microsoft Windows 95. This report gives procedures and examples for the use of the WHAMO program, and includes a complete command reference plus documentation of the algorithms upon which WHAMO is based.



SF 298

Foreword

This study was conducted for Headquarters, U.S. Army Corps of Engineers (HQUSACE) under Civil Works Investigations and Studies Work Unit 32867, "Program 315-Electrical/Mechanical." The Technical Monitor was Andy Wu, CECW-EE.

The work was performed by the Materials Science and Technology Division (FL-M) of the Facilities Technology Laboratory (FL), U.S. Army Construction Engineering Research Laboratories (CERL). The CERL Principal Investigator was Vicki L. Van Blaricum, CECER-FL-M. Dr. Ilker R. Adiguzel is Acting Chief, CECER-FL-M, and L. Michael Golish is Acting Operations Chief, CECER-FL.

Dr. Michael J. O'Connor is the Director of CERL.

Contents

List of Figures and Tables

Figures

Tables

1 Introduction

1.1 Background

Fluid distribution systems and hydropower plants can be severely damaged by water hammer. Water hammer is the forceful slam, bang, or shudder that occurs in pipes when a sudden change in fluid velocity creates a significant change in fluid pressure. Water hammer can destroy turbomachines and cause pipes and penstocks to rupture. Water hammer can be avoided by designing and/or operating these systems such that unfavorable changes in water velocity are minimized.

The WHAMO computer program has been developed to assist engineers in mitigating water hammer by simulating Water Hammer And Mass Oscillation in networks that convey fluids such as water or fuel. Some typical applications for WHAMO include analysis of hydropower plants, pumping facilities, jet fueling systems, and wastewater collection systems. The program determines time varying flow and head (transients) in a network which may include pipes, valves, pumps, turbines, pump-turbines, surge tanks, and junctions arranged in any reasonable configuration. Such transients are generated due to any variation in the operation of a turbomachine or valve within the network, or due to changes in the head or discharge at boundaries of the network. The rotational speed, torque, and power of machines are computed along with the system hydraulic variables.

WHAMGR is an associated graphics program used for creating time history plots from WHAMO simulations. It is also used for plotting turbomachine characteristics input to WHAMO.

The WHAMO Network Builder is an associated program which allows the user to create WHAMO input files using a graphical, point-and-click method to “draw” the network on the computer screen and specify simulation parameters. In previous versions of WHAMO, the user was required to use a complex series of commands to define the network and simulation parameters. WHAMO 3.0 allows either method of data input to be used, and instructions for both are given in this manual.

WHAMO, WHAMGR, and the WHAMO Model Builder run on IBM-compatible personal computers under Microsoft Windows 95.

This manual documents WHAMO version 3.0. Many enhancements and new capabilities have been added since the original release of the mainframe version of the program in 1979-80; however, the fundamental input, computation, and output processing schemes remain largely unchanged. The current Windows-based version of the program is still "input compatible" with the original mainframe version, which means that any input file which successfully ran on previous versions of the program will run on the current version with essentially the same results.

The original WHAMO Program was an outgrowth of and logical extension to previous special purpose programs utilized by the Corps of Engineers. The earliest predecessor program to WHAMO is the MIT-MRD program, developed at the Massachusetts Institute of Technology for the Missouri River Division (Perkins 1964 and 1965). Subsequently, that model was revised and extended to include pumps in order to model the Oahe Power and Pumping Plant for the Omaha District (Resource Analysis, Inc. 1976 and 1978). Over the years that followed, the programs were further modified and enhanced to include more diverse elements and permit their interconnection in any reasonable configuration.

New model capabilities and program modifications made since the original WHAMO release are listed below:

1994-1997

- Ported to MS Windows operating system
- New graphical user interface (GUI)-based WHAMGR for display of program results and characteristic curves of pumps and valves
- New GUI-based WHAMO Network Builder for point-and-click construction of input files
- One-way surge tank
- Turbine startup
- Tabular output improvements including better spreadsheet compatibility, and the ability to specify the number of lines per page and number of decimal places in printed output tables
- Increased array dimensions for machine characteristics

1990-1991

- Ported to DOS operating systems on "IBM-compatible" PCs (including WHAMGR)
- Fluid density may be specified

- Optional output in gallons/minute for discharge and pounds/square inch for pressure
- Pump characteristics may be read from separate files
- Internal branch structure modified to reduce roundoff error on micro-computers
- "Leakage" computed for closed valves reduced by a factor of 10
- New pressure control valve element
- Steady state outflow or overflow from a surge tank
- Starting air chamber surge tank pressures may exceed pipeline pressure

1984

- Variable area surge tanks
- Surge tank outflow option
- Air chamber surge tanks (vented or unvented)
- Pressure actuated valves
- Non-return valves
- Spherical and Howell-Bunger valve characteristics
- Differentiated forward and reverse flow characteristics for valves
- Electric governors
- New governor input options and defaults
- New pump-turbine characteristics input option
- Standard wicket gate cushioned closure option
- Sinusoidal variation optionally superimposed on schedules
- Increased number of system branches and variable diameter conduits allowed
- Graphical plotting of machine characteristics (WHAMGR)

1.2 Objective

The objective of the WHAMO program is to perform dynamic simulation of fluid distribution networks comprising components such as pipes, valves, pumps, turbines, pump-turbines, surge tanks, and junctions. The program calculates time-varying flows, pressures, and heads throughout the network.

1.3 Approach

WHAMO 2.0 for DOS was upgraded to bring it into the Microsoft Windows operating system. A new Windows-based graphics package was developed to enable users to print out or display the program results to any printer or monitor that is supported by Windows. A graphical point-and-click method of data input was developed to replace the old command-line method that was used in previous versions of the program.

1.4 Mode of Technology Transfer

It is anticipated that distribution, support, and maintenance of WHAMO 3.0 will be transferred to the Corps of Engineers Hydroelectric Design Center, CENWP-HDC-P.

2 The WHAMO Modeling System

Before proceeding further, it is important to understand some basics about how WHAMO is structured and how it works. The complete WHAMO modeling system includes three separate programs interconnected via data files. These are the main WHAMO simulation program, the WHAMGR graphics package, and the WHAMO Network Builder.

The main WHAMO simulation program may receive input from either (1) an ASCII text file that has been created with a word processor as described in Section 3.3, or (2) an input file that has been created using the new graphical point-and-click WHAMO Network Builder as described in Section 3.4. Thus, use of the WHAMO Network Builder is optional, but it is much easier and more efficient than using the command language for most users.

The WHAMGR graphics package reads the output file produced by the WHAMO simulation program and allows the user to print the output and/or display it to the computer screen. A variety of display options are available. These will be described later in the manual. WHAMGR can also be used in a pre-processor mode to plot certain input data.

The WHAMO simulation program is the heart of the WHAMO system. It will be described in the sections that follow.

2.1 Introduction to the WHAMO Simulation Program

The main WHAMO simulation program uses an implicit finite difference method for calculating time-varying flows and pressures throughout the network that is being modeled. Details are in Chapter 6. Numerical techniques used to approximate partial differential equations such as the water hammer equations can be generally classified as implicit or explicit. Implicit methods generally require simultaneous solution of a set of equations while explicit formulations can be solved directly. Explicit methods are constrained according to system geometry to work at computational time steps which are typically very small, while the implicit methods are not so constrained. Implicit methods require greater computational effort per time step because of the necessity of solving simultaneous equations. The amount of effort required for a simulation can be reduced by varying the length of the time step during the simulation. WHAMO allows this. Any initial, high frequency water hammer response in a system should be modeled with short time steps, but in simulations where the water hammer dissipates and mass oscillation becomes predominant, the time step can be greatly increased during the simulation with no significant loss of accuracy.

Implicit, finite difference techniques, such as used in WHAMO, average time derivatives over conduit reaches and time steps. Therefore, it might be expected that the solution would be less accurate at sharp wave fronts. The implicit solution technique is, in fact, able to accurately model very sharp wave fronts (Perkins 1963). Perkins showed that results obtained with implicit method simulations of rapid and instantaneous valve closure in a pipe agreed favorably with results obtained by the explicit method of characteristics. In some cases, however, a spurious, minor oscillation can be superimposed on simulated solutions in the vicinity of particularly sharp transients resulting from pump station power failure or very rapid valve closure in pipelines. This noise damps out quickly without altering the overall solution and can sometimes be eliminated or minimized by careful selection of time steps ($\Delta t = n * \Delta x / c$) during the period of sharp transients (Perkins 1963).

The WHAMO simulation program is constructed in a modular fashion to make it more general, ease the addition of new capabilities, and produce a more efficient code and storage structure. This is transparent to the user. The following basic elements form the program modules:

1. A set of building block subroutines which represent the one-dimensional continuity and momentum equations for compressible liquid flow in closed conduits.

2. A set of building block routines which represent the head discharge relations across typical elements such as turbomachines, valves, and junctions.

3. A set of input routines which accept a description of the preceding elements and their interconnection.

4. A set of organizational routines which restructure the input data into a matrix form representative of the system to be modeled, and a routine for solving the resulting equations.

5. A set of computational routines which automatically generate an initial steady-state condition for the system, consistent with input boundary conditions.

6. A set of output routines which allows the program user to specify the type, amount, and form of computational results to be produced. Output may be printed in either a tabular or graphical form, exported to a spreadsheet, or saved as an ASCII text file. Where simulations are long, time steps short, or the system complex, the program produces a huge amount of output, therefore the user may select the locations and parameters for which an output history is desired.

WHAMO follows certain basic steps during a simulation run. These are:

1. Read and check input data — The user's commands and associated data are read in. The data are processed as necessary and checked for errors.

2 Build system connectivity — The user's specifications of the system structure are reinterpreted to a form compatible with computation. The resulting system is checked to insure that it is physically reasonable.

3. Display input data and system structure — All data input as well as the system structure and other information determined by the program are printed in tabular form. This allows easy verification of the input data by the user.

4. Steady state generation — An initial, steady state condition compatible with the specified boundary and operating conditions is determined for the system.

5. Transient simulation — System response to specified transient machine operation or boundary condition is simulated. Computational difficulties are monitored by the program's self-checking algorithms.

6. Output — The simulation results are printed or stored for later processing according to the user's instructions.

The WHAMO program considers a hydraulic system as constructed of a number of individual components of various types, interconnected in a particular network configuration. This information must naturally be supplied to the program by the user. The model further requires specification of system boundary conditions and operating conditions for the simulation period. These data are interpreted and processed by the program and computations are performed to simulate the transient state hydraulics of the system.

Various aspects of the system specification by the program user, the interpretation and processing of these data by the model, and the computational strategy employed in the hydraulic simulation are introduced in the following sections. The intent is to acquaint the user in a general way with the modeling operation. The details of program input and computational algorithms are presented in subsequent sections of this manual.

2.2 WHAMO System Elements

The basic physical elements considered by WHAMO fall into four main categories: flow elements, turbomachines, boundary elements, and junctions. Altogether, nearly 20 specific element types are available to the user for construction of a system.

For each element type, the model employs special forms of mathematical equations to represent the hydraulics of a given component. These equations are utilized in pairs, one based on the continuity principle and one based on the momentum (or energy) conservation principle as applied to a particular element. These are described in Chapter 6.

When the user specifies an element of a particular type, he must designate a name by which it can be referenced and supply data on the properties which are relevant to the hydraulics or mechanics of that element. For example, the length, diameter, roughness, and celerity of a conduit must be specified. Simply stated, enough data must be supplied for an element that continuity and momentum equations can be written under all conditions to be simulated. Input data requirements are presented in detail in Chapter 4.

2.2.1 Flow Elements

2.2.1.1 Conduits. The conduit element is a closed conveyance element of circular cross-section whose hydraulic performance can be expressed in terms of the one-dimensional continuity and momentum equations for compressible liquid flow. Total head, pressure head, piezometric head, and discharge are computed at the conduit element end points. For computational reasons, a conduit element may include a number of segments.

2.2.1.2 Variable Diameter Conduits. These are conduits identical to those above, except in the respect that the conduit diameter varies along the axis of the conduit.

2.2.1.3 Dummy Elements. Not a physically realistic element, dummy elements are simply a connector supplied for user flexibility in constructing systems.

2.2.1.4 Control Valves. Control valve elements are applicable to throttling devices for which a discharge coefficient or head loss coefficient is specified versus valve opening, and a schedule of valve openings is supplied. Characteristics for butterfly, gate, spherical, and Howell-Bunger valves are supplied by the program. The user supplies characteristics for other types of valves.

2.2.1.5 Pressure Control Valves. The opening of these valve elements is determined according to user-specified functions of pressure at one or two reference points in the system, instead of a predetermined schedule.

2.2.1.6 Non-Return Valves. These elements allow flow in one direction only. A single head loss coefficient may be applied to flow in the allowed direction.

2.2.1.7 Expansions and Contractions. These elements are used to model the head losses resulting from abrupt changes in conduit diameter within a pipeline.

2.2.2 Turbomachines

2.2.2.1 Pumps. Pump elements represent hydraulic machinery which can add energy to the hydraulic system using mechanical energy supplied externally. The hydraulic performance of the machinery is represented through performance data which consists of dimensionless tables of head and torque versus speed and discharge through all four quadrants of pump and turbine direction performance data. The input also includes specification of the operating mode of each pump. Output includes pump speed, torque, discharge, and head difference.

2.2.2.2 Turbines. The turbine element models turbomachinery operating with flow in the turbine direction only which can remove energy from the hydraulic system. The hydraulic performance is expressed by input tables in either of two forms. The tables may be of prototype data consisting of discharge and efficiency as functions of wicket gate opening and head across the turbine. More usually, homologous model data is supplied, in which model discharge and power are given as functions of the gate opening and peripheral velocity factor (phi). In either case, the speed, discharge, gate opening, power production, and head across the turbine are calculated. In addition, the governor element (see below) is available for use with the turbine element to model automatic governance of the turbine under small changes in speed or load.

2.2.2.3 Pump-Turbine. The pump-turbine element represents hydraulic machinery designed to operate in both pumping and turbine mode for pumped storage-power generation and similar operations. The hydraulic performance of the machinery is represented through model performance data which consists of tables of discharge and torque vs. speed and gate opening under unit head conditions. When in the turbine mode, automatic governance of the machinery may be modeled using governor element (see below). The pumping mode operations are controlled through input histories of gate position versus time rather than by modeling any automatic control mechanisms which may exist. The program calculates speed, torque, power, head difference, discharge, and gate opening for output purposes.

2.2.2.4 Turbine Governors. Turbine governor elements control the operation of turbines and pump-turbines under small changes in speed or load. The governing algorithms are not applicable to large changes, such as full load rejection, since the limits, discontinuities, and non-linearities of the actual governing mechanisms are not modeled by the program element.

2.2.3 Boundary Elements

2.2.3.1 Surge Tanks. Simple and differential surge tanks may be modeled by the surge tank element. The effect of restricted entrance orifices may be considered as desired. Output for this element includes the water surface elevation in the tank, as well as discharge.

2.2.3.2 One Way Surge Tanks. One way surge tank elements allow the surge tank water surface to be below the hydraulic grade of the pipeline. They must be separated from the pipeline by a non-return valve.

2.2.3.3 Air Chambers. Pressurized air chamber surge tanks may be modeled. Restricted air inflow and outflow to the air chamber may be simulated.

2.2.3.4 Free-Surface Boundaries. Boundary water surface elevations, such as at reservoirs, are specified through this element either as constants or as functions of time.

2.2.3.5 Flow Boundary. Flow boundary elements specify discharge boundaries to the system either as constant or time-varying flows.

2.2.4 Junctions

2.2.4.1 Simple Junctions. Junctions of six or fewer elements may be modeled as simple junctions. Such junctions consider the conservation of flow, but do not compute head loss due to the junction.

2.2.4.2 Tee Junctions. Junctions of three elements are considered by the Tee Junction element, which computes the loss of head using the relations developed by Gardel (1955).

2.3 System Construction

The interconnection of the individual elements must be completely defined for a system in order for simulation to be possible. This information is supplied by the program user as part of the data input. Based on the configuration in which the elements are assembled, the program can organize the individual governing equations which represent the various elements into a unified set of equations written in terms of common variables which can be solved simultaneously to accurately represent the hydraulics of a system.

The basic building blocks by which the user constructs a system are nodes and elements. A node represents a point of the system at which two or more elements are joined or at which boundary conditions (or diameter changes) are located. The nodes of a system are linked together by elements and every element links two nodes, except for boundary conditions and diameter changes. Nodes which are linked by three or more elements receive special treatment. These junction nodes are in themselves elements, either simple junctions or Tee junctions. The connectivity input then consists of a series of statements which list each element by name along with the designation of the two nodes which it links, or the single node at which the element is located.

With this input from the user, the program prepares a list of elements in order of their appearance in the connectivity input statements. This list is referred to as the Computational Element Sequence (CES) to distinguish it from another element list which is formed as the component elements are individually defined and their properties specified. This second list is called the Defined Element List. Each element in the Defined Element List is unique, but may appear more than once in the Computational Element Sequence. Subsequently, the CES is reordered such that elements linked to the same node are listed consecutively, in upstream to downstream order, where the flow convention is defined by the user in the syntax of the input statements. Associated with the CES list, the program stores the upstream and downstream node bounding each element as well as cross references to the Defined Element List.

After completion of all program input, a coherent description of the system and its components is constructed. At this point, the program must introduce some internal definitions of the system which are more consistent with the mathematical formulation of the problem than the external representation supplied by the user. In particular, the external node system defined by the user is duplicated and expanded into an internal node system, and the system is separated into branches.

The internal node system is constructed so that all points between which head and continuity equations will be solved are represented. For example, a Surge Tank Boundary condition is represented by a single external node in the SYSTEM input command. However, for mathematical purposes at least three internal nodes are defined: one internal node where the surge tank joins the preceding element, one at the free surface, and at least one more to break the tank into two segments — an infinitesimal segment over which an entrance loss can be calculated and a finite segment over which head and discharge variations from the bottom of the tank to the free surface are calculated. At least four internal nodes are defined for differential surge tanks. Similarly, expansions and contractions require the addition of one internal node; conduits require the addition of one internal node for each segment after the first; and junctions require an additional internal node for each pipe joined at the junction after the first. The system of internal nodes are cross-referenced to the external nodes through a number of external node arrays. Thus, all references by the program user may be in terms of external nodes only.

The identification of branches is also accomplished in this process. Originally, for the purposes of the computation, a branch was defined as a series of linearly connected elements with endpoints that are either boundary conditions or junctions. For implementation on PCS, branch definition has been modified due to numerical precision limitations such that each specified conduit is assigned to a separate computational branch. The elements within the branch may be conduits, pumps, valves, turbines or diameter changes. Such branches are distinguished by the convenient property of being reducible to a single set of momentum and continuity equations. The reduction process will be explained in Sections 2.6 and 6.7.

With the branch structure, element structure, and internal node structure of a system so defined, the program now has a logical basis with which to structure a system matrix (plus associated right hand side vector) organizing the numerous

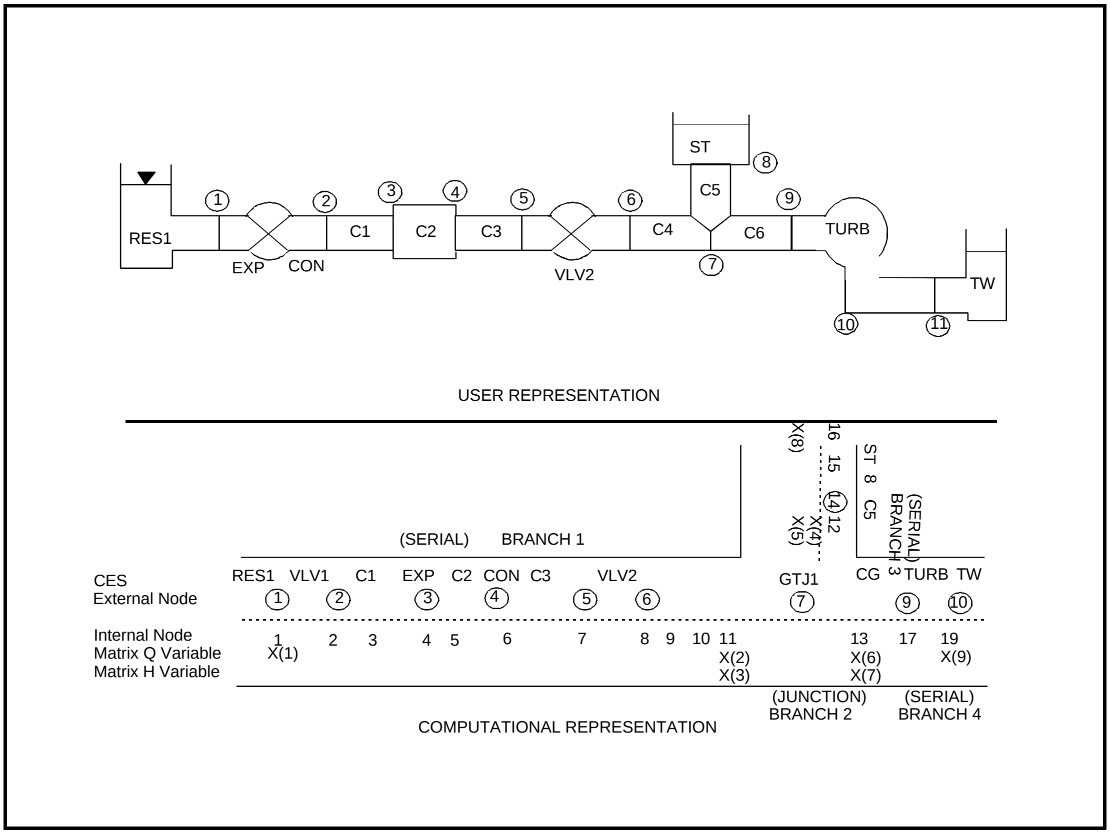


Figure 1. System connectivity.

element equations into a unified, comprehensive set which can be solved simultaneously in an efficient manner.

Figure 1 summarizes the connectivity of a hypothetical example system, first from the point of view of a user preparing connectivity input and then from a computational point of view after this input has been processed by the program. The designated matrix Q and H variables shown in the figure will be referred to in later sections of this manual. It is worth noting at this point that one of the prime ends of the connectivity process is to determine which variables are included in the system matrix.

2.4 System Boundary Conditions

At the boundaries, or end points of a system only certain types of elements may be specified. These are the boundary elements introduced in Section 2.2: reservoir elements, flow boundary elements, or surge tank elements. These provide information to the model which is essential to the solution of any and all of the system equations. This information consists of the head value, discharge value, or a function relating head and discharge at each boundary point. Additionally, boundary condition elements signal the program to end an old branch or start a new branch during connectivity processing (Section 2.3).

The reservoir element is used to assign a specified head value (or schedule of values) at a boundary. This is representative of a headwater condition, tailwater condition, or free outfall at the end of a system. This will probably be the most commonly used boundary element.

The flow boundary element is used to assign a specified discharge value (or schedule of values) at a boundary. This element does not have a simple physical analogy. It is very useful, however, for representing a connecting portion of a system not to be explicitly modeled. For example, one branch of a system made up of parallel generating units might be modeled explicitly while the others, including penstocks, surge tanks, and turbines etc., are represented with a constant discharge. This saves both data preparation and computational expense.

Where a surge tank element occurs at the end of a branch, a function relating the head and discharge at the boundary is implicitly defined. See Chapter 6 for elaboration on the nature of this function.

2.5 System Operating Conditions

For any system which includes a pump, turbine, pump-turbine, or valve, instructions specifying the operation of these components must be part of the input data. A variety of common modes of operation of these elements can be simulated with WHAMO. These are outlined briefly below.

Valve operation must be specified explicitly for each valve in a system by the use of a schedule of position (in terms of percent opening or angle) versus time. For pressure control valves, valve position is determined according to a specified function of pressure at one or two reference points in the system. Obviously, the equation to be written across a valve depends significantly on the valve opening.

Three options for pump operation are available:

1. Constant synchronous speed pumping in which as much power is drawn as is necessary to maintain speed.

2. Synchronous operation until power shut-off, after which no power reaches the pump and speed is allowed to vary according to the mechanics and hydraulics of the situation.

3. An "off" option which is equivalent to not having the pump in the system at all.

A turbine may be operated in one of five modes:

1. Synchronous speed generation in which the generator power output is equal to the hydraulic power to the turbine minus losses.

2. Synchronous generation until load rejection, after which the generator load on the turbine is zero and the turbine speed is allowed to vary according to the hydraulic and mechanical conditions.

3. Turbine startup, in which turbine acceleration from 0 rpm under non-governor wicket gate control is simulated.

4. Governor controlled generation, in which a schedule of generator load is specified and turbine speed and wicket gate position vary according to a specified governor function.

5. An "off" option which is equivalent to replacing the turbine with a closed valve.

For the first three operating modes above the user must supply a complete schedule of wicket gate openings, while for the fourth option, as stated, a schedule of generator loads and a specification of governor properties is necessary to the simulation.

The options listed above for pumps and turbines are all available for pump-turbines. For pump-turbines, a wicket gate schedule must be supplied for the pump and pump shut-off options as well as for the generate, startup and load rejection options.

Complete specification of system boundary conditions and operating conditions is clearly a prerequisite to a hydraulic simulation. Furthermore, these are the only means by which transients can be generated in WHAMO. Unsteady hydraulic response will result from varying conditions at the boundaries of the system, from varying the opening of valves or wicket gates, and from changes in machine operation, e.g., pump shut-off, turbine load rejection, or load variation under governor control.

2.6 Computational Procedure

The computational strategy adopted for the transient simulation is briefly introduced in this section. The main dependent variables of interest are the total head, H, and discharge, Q, at each internal node of the system. Beginning with time 0 (zero) the computation proceeds from time step to time step to the end of the simulation. A complete solution of the system is computed at each time before proceeding to the next time step. The length of each time step and the total length of the simulation is specified by the user.

Between each pair of linked internal nodes, two governing equations can be written in terms of the Hs and Qs at the nodes. These equations are based on momentum or energy conservation and continuity principles. The exact form of the equations depends on the type of element which links the two nodes. If the equations are differential equations, a finite difference approximation is formulated, and if non-linear, a linear approximation is formulated. The linear finite difference equations written between each pair of linked internal nodes are termed the inner equations. At each time step these equations are determined anew, depending on the properties, operating mode (where applicable) and past history of each element.

It can be seen that two equations in four unknowns can be written for each segment (that portion of a system bounded by two linked nodes). For each branch of N segments there are a total of 2N inner equations and 2(N+l) unknowns, one H and one Q at each of the N+l internal nodes contained in the branch. The deficiency of two equations will be made up by the introduction of appropriate boundary and/or junction equations at the ends of the branch.

The set of 2N equations in 2N+2 unknowns for each branch is reduced for reasons of economy to a set of 2 equations in 4 unknowns by a recursive procedure called forward construction. The procedure is described in Section 6.7 and in Resource Analysis, Inc. (1976). The two branch equations which result from forward construction are termed the outer equations, for these are written in terms of only the Hs and Qs at the ends of the branch.

The branch outer equations of a system can now be solved simultaneously with the junction equations and boundary equations to determine H and Q at all the branch end points. In fact, instead of adding boundary equations to the set, unknowns are removed from the set by inserting the known boundary H or Q values into the outer equations where those variables appear. (Note: Subsequently in this manual, junctions will be considered branches in themselves, and junction equations will be considered outer equations.)

The set of outer equations is organized into matrix form (referred to hereafter as the system matrix, including both the coefficient matrix and right hand side vector) for solution. The structure of this matrix was formed as part of the connectivity process (Section 2.3). Referring back to Figure 1, note that those variables which will be included in the system matrix are predetermined from the connectivity and numbered. At the same time, matrix row numbers are assigned to each branch for the outer equations which will be written for that branch.

A standard, "off the shelf" matrix solver is utilized to solve the set of equations. This routine uses the Gauss elimination technique with complete pivoting on the largest element. This is often the most computationally costly part of the process. The cost of solving an n-dimensional matrix is proportional to a factor between n^2 and n^3. The branch reduction process was therefore adopted to minimize the size of the system matrix for the sake of economy. In contrast to matrix solution, the cost of branch reduction is roughly proportioned to the number of inner equations to the first power.

After H and Q at the branch end points have been calculated by simultaneous solution of the outer equations, a process of back substitution uses the end values to determine the H and Q at the intermediate nodes of a branch. This process is outlined in Section 6.9 and in Resource Analysis, Inc. (1976). Essentially, it is the reverse of the forward construction process.

Finally, in a separate computation at the end of each time step the mechanical speed, power, and torque at machines is determined. The model is now ready to repeat the entire process, beginning with forward construction, at the next time step.

2.7 Initial Steady State Conditions

Prior to the transient simulation, the initial steady state heads and discharges must be determined for the entire system. The initial steady state is generated automatically by the model for the boundary conditions and the machine and valve operating conditions specified. Computational procedure used for the steady state determination is almost exactly that of the transient simulation, with the following modifications.

First, the program makes an initial guess of the discharge in each branch. This is necessary to form the inner equations for most element types. This guess, if not supplied by the user is made on the basis of boundary conditions at the ends of branches, the operating characteristics of machines which are part of branches, and continuity relations at the junctions.

Another steady state modification is that the governing equations for conduits are altered to eliminate unsteady and compressible terms and the governing equations for surge tanks are altered to allow no flow (see Chapter 6). Otherwise, the development of inner equations, forward construction, matrix solution, and back substitution proceed as described in the preceding section. The final distinction is that the steady state computation is not a repetitive process proceeding from time step to time step, but an iterative process which is cycled until convergence to a solution. This convergence is achieved when the discharge estimates used at the beginning of a computational iteration agree closely with the calculated discharges at the end of an iteration. If they do not agree, the calculated discharges from the old iteration are used as the estimates for the new iteration and the computation is begun anew. If, after a given number of cycles, convergence is not achieved, the program will request that new discharge estimates be supplied by the user. This is not a common circumstance. However, the user always has the option of specifying estimates of branch discharges, machine operating heads and wicket gate positions in order to speed the convergence process.

3 How To Use WHAMO

This section is a detailed guide to the application of the WHAMO model to hydraulic transient simulation. It is designed to guide the user through the entire process of performing a WHAMO simulation. The flow chart in Figure 2 shows the sequence of steps to be followed, along with a reference to the corresponding section in this manual.

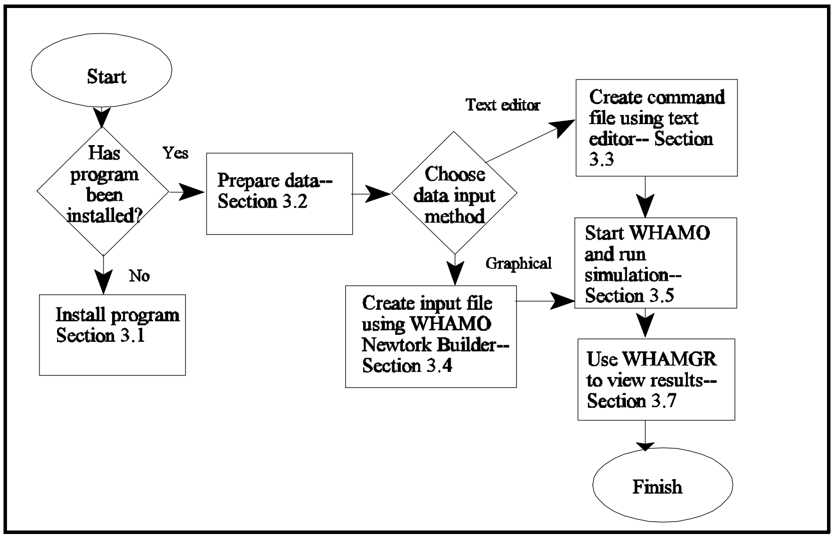


Figure 2. Procedure for using the WHAMO simulation system.

Please note that there are two possible methods of creating a WHAMO input file. They are:

- Command file method (old method)
- WHAMO Network Builder (new method).

Experienced WHAMO users may feel more comfortable using the command file method, in which a word processor or text editor is used to create an input file as explained in Section 3.3.. This is the method that has been used in previous versions of WHAMO. It requires the user to be familiar with the WHAMO command language. The other method is to use the WHAMO Network Builder as described in Section 3.4. The Network Builder allows the user to create an input file by drawing the network graphically on the computer screen. New users may prefer this method because familiarity with the WHAMO command language syntax is not required. However, users of both methods will want to refer to the command glossary in Chapter 4 for detailed information on the data that is required.

3.1 Installing the WHAMO System

The required computer configuration is as follows:

- IBM-compatible PC
- Windows 95 operating system
- 8 Mb RAM minimum
- 7 Mb available disk space minimum.

Because WHAMGR and BUILDER are Windows programs, they can be used with any true Windows-compliant display (monitor) or printer.

You will need to install the main WHAMO program (WHAMO.EXE), the WHAMGR program, and the WHAMO Network Builder.

First, create a new directory (folder) on your computer's hard drive for the main WHAMO program. (If you do not know how to create a new directory, please refer to your Windows 95 user's manual.) It is recommended that you name the new directory C:\WHAMO, but the directory can be given any name you wish. To install the main WHAMO program, copy the compressed file "WHAMOSIM.ZIP" from the floppy disk labeled "WHAMO" to the directory you have just created. Decompress the file with WINZIP or a similar decompression utility.

The WHAMGR and WHAMO Network Builder installation disks each contain a file called SETUP.EXE that launches the Windows 95 setup procedure. The default directory for WHAMGR is C:\Program Files\WHAMGR. The default directory for the Network Builder is C:\Program Files\Builder. You may change these directory names or locations during the setup procedure if you wish; some users may wish to install all of the programs in the same directory (e.g., C:\WHAMO). Follow these steps to install each application.

1. Insert Disk #1 and open the folder for the A:\ drive
2. Double-click on the SETUP.EXE icon
3. Follow the on-screen instructions to load the program. Insert any additional disks when prompted.

VERY IMPORTANT: After you have completed the initial setup, you must register the system file named “vcf132.ocx” with Windows. If you skip this step, the Network Builder and WHAMGR will not operate properly! Use the following steps to register this system file:

1. Open an MS-DOS window. After the prompt, change directories to C:\Windows\System. (If you do not know how to do this, please consult your Windows 95 documentation.)

2. After the DOS prompt, key in and enter the following command:

   C:\Windows\System> **regsvr32 vcf132.ocx**

   Type in the character string exactly as it appears in bold letters above.

3.2 Data Preparation

The basic steps to preparing WHAMO input data for simulation of a hydraulic system are outlined below, followed by discussion of further input options and modeling issues. The terminology in the following steps is directed at the user who is utilizing the command file method of creating input. However, the same steps still apply to a user who is utilizing the WHAMO Network Builder method to create the file. Instead of writing and inputting commands, the Network Builder user will be using the mouse to place system elements on the screen and will be using the pull-down menus to enter data about the system elements and the simulation parameters. This should be kept in mind when reading this section.

The steps are:

1. Data gathering
2. Preparation of system schematic diagram
3. Listing of elements leading to preparation of the SYSTEM command
4. Input of element properties data
5. Organization and input of machine and valve characteristics tables
6. Definition of system operation
7. Specification of output requests
8. Specification of computational parameters
9. Adding execution control statements to complete the input file
10. Review and verification of input data

11. Updating input data for subsequent simulations.

In detail, the steps are:

1. The first step is to assemble all of the available data related to the hydraulics of the system under consideration. The source of this data will vary depending on whether an existing system, or the proposed design of a new system, is being analyzed. The full extent and exact nature of the data required for WHAMO modeling will become clear as the user progresses through this manual. The first requirement is a sketch, drawing, or set of drawings which depicts the pipe geometry and location of all key components of the system.

2. A simple schematic diagram is prepared to represent the system configuration in the same terms as the WHAMO model input; that is, as a series of individual elements which are joined at uniquely numbered node points (see Figure 3). Each individual element is a component of a type introduced in Section 2.2: conduit, valve, turbo-machine, surge tank, diameter change, junction or boundary condition. Except for tapered conduits, a conduit element has constant properties along its length. Where the hydraulic properties change, a new conduit element is created and joined to the preceding one by a node. Also, since head and discharge output is only available at nodes, when the user is interested in getting this output at intermediate points of a conduit, he must place nodes at those points and break the conduit into individual elements which link the nodes together. Intakes, bends, and orifices, etc., where special losses are calculated for a conduit need not be individual elements but are modeled as part of the conduit element. Wicket gates and governors that control turbines and pump-turbines should not be shown as individual elements but are considered part of the machine they serve.

   As described in Section 2.4, appropriate boundary elements must be located at the end points of the system. Boundary elements cannot form an entire branch, so that they may not be joined directly to another boundary element or a junction. For example, where a surge tank connects directly to a junction, the user must show an intervening dummy conduit in the schematic. Further details on the definition of system elements can be found in Chapter 4.

   Node points are placed where elements join, and at the ends of the system. Nodes serve as locations by which the user can interpret output data and by which the program can determine the interconnection of elements (See Section 2.3). Junctions, boundary elements (including surge tanks), and diameter changes are located at nodes, in contrast to all the other element types which link between two nodes. Therefore, at a junction, all branching pipes are linked to the same node — the node at which the junction is located. Similarly, a node is not placed between a boundary element and the next element of the branch. The node at which the boundary element is located is the upstream or downstream node to which the next element is connected. Neither are nodes placed between a diameter change element and the pipes upstream and downstream. One node serves as the location of the diameter change and the connecting point of the conduit elements.

Each node and each element in the schematic diagram must be given a unique designation — a four-character alphanumeric name for elements and an integer number, one to four digits, for nodes. This includes all the element types with a single exception, simple junctions. (Tee junctions must be named.)

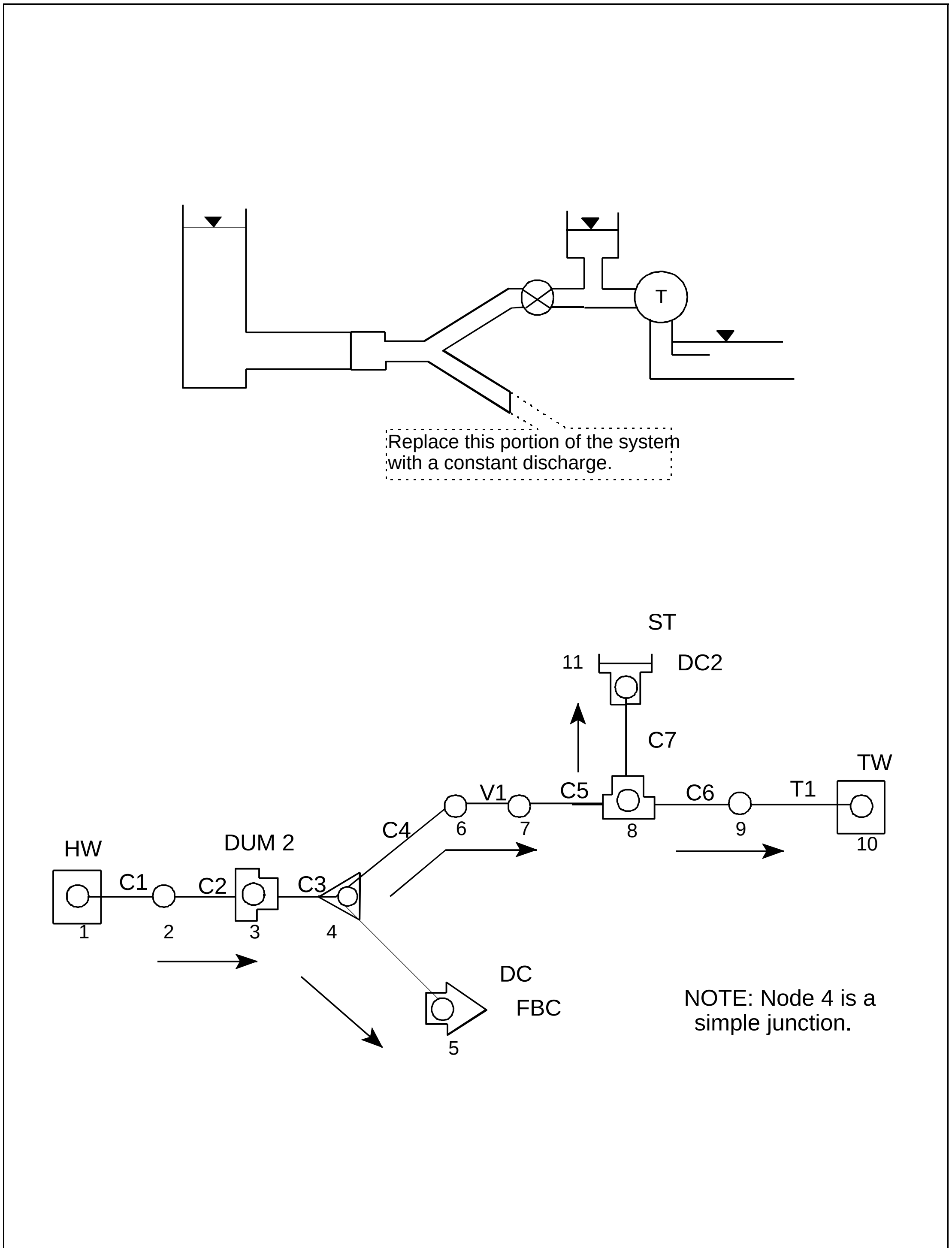


Figure 3. Schematic diagram of hypothetical system.

The final task in preparing the schematic is to designate the direction of positive flow for each branch of the system. A branch is defined in Section 2.3 as a series of linearly connected elements with endpoints that are either boundary conditions or junctions. The choice of positive flow direction is arbitrary unless a turbo-machine or surge tank is included in a branch. A surge tank must be at the downstream end of a branch, while positive flow must be in the pumping direction for a pump, and in the turbine (or generating) direction where a turbine or pump-turbine is part of a branch. A sample schematic diagram for a hypothetical system is shown in Figure 3.

3. From the schematic diagram, a list is to be made of all the component elements of a system. Each simple junction should be included in this list even though these are not named. Beside each element list the upstream and downstream node, or the node location for B.C.'s, junctions, and diameter changes, and the element type. An example of the element list for the system of Figure 3 is shown in Table 1. This list, plus the schematic, is the key to preparing model input and interpreting the output. The SYSTEM command, used to input system connectivity, can be written directly from the element list (Chapter 4).

4. The next step is to go through the list element by element, excepting only simple junctions, preparing the input data specifying the properties of each using the appropriate primary command for the element type (e.g., CONDUIT, PUMP, etc.). Details of the data required and the commands for each element type are included in Chapter 4. It will be noted when writing VALVE, RESERVOIR, and FLOWBC commands, that operating schedules may be required. These may be omitted at this stage of the process and taken up later.

   If a turbine or pump-turbine is to be simulated under governor control, the governor properties should be input for that element at this time using the GOVERNOR command. The VALVE, RELIEF, PUMP, TURBINE, and P-T commands require specifications of a characteristics type. Machine characteristics are dealt with in the next step.

Table 1. Element list for hypothetical system.

Element I.D.	Type	Node Location	Upstream Node	Downstream Node
HW	Head Boundary	1-	—	—
Cl	Conduit	—	1	2
C2	Conduit	—	2	3
DC	Diameter Change	3	—	—
C3	Conduit	—	3-	4
—	Simple Junction	4	—	—
DUM	Dummy Conduit	4	5	—
FBC	Flow Boundary	5	—	—
C4	Conduit	—	4	6
V1	Valve	—	6	7
C5	Conduit	—	7	8
TJ	Tee Junction	8	—	—
C6	Conduit	—	8	9
Tl	Turbine	—	9	10
TW	Head Boundary	10	—	—
C7	Conduit	—	8	11
DC2	Diameter Change	11	—	—
ST	Surge Tank	11	—	—

5. Characteristics tables for machines and valves are input not for a particular element (using, say, the PUMP command), but separately so that one set of characteristics can apply to two or more similar or homologous elements without having to be input more than once. (For example, characteristics for a pump are input using a separate PCHAR command.) The input data for each machine or valve element must include a reference to a particular characteristics table (or TYPE). Characteristics tables are input according to the directions in Chapter 4. The form that these tables must take is specified for each machine type. If the data available for a machine or valve does not fit one of the forms allowable for that type, then the data must be altered using appropriate similarity or physical relations. It may sometimes be possible to avoid or minimize this task by designating a pump or turbine unit as a pump-turbine. In this way, the pump-turbine form of the characteristics may be utilized if more convenient without altering the simulation options.

For a pump, fictitious characteristics for "wicket gate" values other than 100 will have to be input.

6. At this point, the physical properties and configuration of a system are completely defined. The remaining input relates to the particular simulation to be performed for the system defined above.

   As outlined in Section 2.5, the operation of each machine and valve of the system must be specified for the simulation. The options available for machines are described in Section 2.5 and Chapter 4. These are input using commands such as OPPUMP, referring to a particular pump element. The OPTURB and OPPT commands require specification of a schedule number as well as the operating mode of a particular element. The schedule number refers to a wicket gate schedule or load schedule (depending on the operating mode) input separately using the SCHEDULE command. Thus, to specify load rejection for a turbine, the statement REJECT will appear along with a schedule number under the OPTURB command, and the wicket gate schedule will appear separately under a SCHEDULE command with an identifying number the same as that designated under OPTURB.

   The operating schedules for valves and/or boundary elements (not including surge tanks) with time varying conditions are also input using the SCHEDULE command. It will be necessary at this point to either go back to the original VALVE command for each valve element to add the schedule number which applies or, possibly to enter additional VALVE commands which include only the schedule number and element identification. This applies to boundary elements as well as to valves.

7. The physical system and its operation have now been completely defined. Next, the user decides what variables are of interest for this simulation and specifies that the simulated time histories of these variables be printed or saved for plotting. The HISTORY, PLOT, or SPREADSHEET command is utilized. In addition, if displays of input data are desired, they can be requested using the DISPLAY command. There is also the SNAPSHOT output option. With this option, the user requests printing of the heads and discharges for the entire system at specified times.

8. The computational parameters are input using the CONTROL command. These include the computational time step and output interval to be used for different portions of the simulation, and the total length of the simulation. This data must be included even if a check run or initial conditions run is

being requested. A computational weighting factor (THETA) may also be specified as described in Section 4.4. The default value for THETA equal to 0.6 is usually suitable so this input may be omitted. The computational time step selected for portions of a simulation when water hammer is significant is typically of the order of 0.1 seconds. When the slower, more gradual mass oscillation predominates, time steps as long as 10 seconds may be appropriate. In general, the nature of the system and the simulation, along with the experience of the user, will dictate the time step lengths selected.

9. The input file is now almost complete. A title line must be added to the front of the file. This first line is not processed as the other input, but is simply read as alphanumeric input. It is output in all header boxes and identifies the output tables and the plot files. Following the input data, a GO command causes the input editor to stop processing input data and to start execution. The command following GO must be either GOODBYE, NEWRUN, or RERUN. GOODBYE terminates execution, NEWRUN introduces a complete new input stream for a new simulation, and RERUN is used to modify certain items of the system or the simulation specifications, and then re-start the simulation. The RERUN option should be used sparingly for it can lead to wasted simulations. (Please note: users of the WHAMO Network Builder do not need to perform this step.)

10. Prior to performing actual simulations, it is advisable to first do a CHECK run with all display tables printed. When a CHECK command is included in the input data, the program processes the data, checks for logic errors, and displays the data, but does not execute the simulation. The user can then review the displays and check for errors in the input data. When the input processor detects logic errors in the input data, the run automatically becomes a CHECK run. After the input data has been corrected and verified, the user may choose to turn off some or all of the input displays in subsequent runs.

11. After the input data has been prepared, reviewed, and corrected, and the first simulation successfully completed, the user will probably want to do additional simulations with the same system, altering somewhat the system operation or physical properties. The nature of the free format input makes it easy to update the input data for a new run. Essentially, if a particular data item is specified more than once in the input, the program uses the data as it last appears. Therefore, changes in element properties can be effected by either repeating or replacing the applicable element command. The operating mode and schedule number for an element can be changed in the same manner. Numerous schedules can be included with the input data, with the operation

commands referring only to those schedules which apply for a given simulation. In fact, numerous elements can be defined in the input, while only those included in the SYSTEM command will be included for that simulation. The above examples are only some of the ways in which the user can take advantage of the free format input language. The use of the RERUN command further allows different simulations to be performed in the same run.

3.2.1 Further Input Options and Modeling Issues

1. In addition to CHECK runs and full transient simulation runs, the user may request computation of only initial steady state conditions. The model computes these automatically as part of every simulation, but if an IONLY command is included with the input, output is printed after the steady state determination and execution is terminated. Two of the possible uses of the IONLY option are these. (a) The user wants to replace a portion of a system with a flow boundary element in order to save computational expense. In order to determine the constant discharge value which he should use at the flow boundary, he first specifies an IONLY run for the complete system. (b) The user wants to simulate a generating system with a turbine under governor control beginning (and/or ending) at a particular wicket gate position. He needs to know the initial load to specify which corresponds to that wicket gate position. He first does an IONLY run with the turbine operating in the GENERATE mode, the wicket gate set to the desired value, and the turbine power specified as output. There are many other possible applications of the IONLY option. Obviously, time history plots cannot be made from IONLY runs, but the SNAPSHOT output option is available.

2. Although it was not suggested in the step-by-step procedure, it is possible for the same element with the same identifying name and properties to be included in more than one location in the system. This short cut is not recommended in general because it tends to be confusing. It is prohibited for machines, valves, and differential surge tanks. A better shortcut to use when two or more elements in a system have identical properties is to define AS command to indicate that the properties of one are identical to those of another. When two elements have close, but not identical properties, the AS command can still be used, followed by the value of the distinguishing property(s).

3. The computational time of WHAMO modeling is a function primarily of the complexity of the system to be simulated and the number of simulation time

intervals. The latter factor is quite straight-forward. After subtracting the time (which is not great) of loading the program into memory and processing the input data, the simulation cost is roughly proportional to the total number of time steps requested, which is to say, the number of times the computational process is repeated. It therefore behooves the program user to limit the length of a simulation to the time period of particular interest and to choose time steps as long as computational accuracy will permit. Advantage should be taken of the WHAMO capability to vary the length of the time steps during the simulation period, allowing short time steps while water hammer predominates and long time steps during mass oscillation. System complexity increases the computational cost per time step. To a lesser extent, this cost increases as the number of system elements and conduit segments increase. The more important factor controlling computational cost is the number of branches in the system, including both junction branches and serial branches, for this dictates the size of the matrix to be inverted at each time step. Time spent doing matrix solution, usually the most time consuming part of the computation, is a function of the dimension of the matrix raised to the second to third power. It is for this reason that substantial savings can sometimes be achieved by replacing a complex portion of a system with a flow boundary element when that portion of the system is not under study. Additionally, judicious representation of the system can sometimes minimize the number of branches. Figure 4 shows schematically a pipe flow splitting in four directions. If modeled exactly as constructed, ten system branches are included. A slightly modified version reduces the total to six.

4. Truncation error, or the error inherent in approximating the governing differential equations using discrete intervals of time and segments of space, is reduced as the time intervals and spatial segments are diminished in size. Accordingly, at some cost in computational expense, the user should expect greater computational accuracy using smaller time steps and finer segmentation of long conduits. Ideally, these would be reduced until there is an insignificant change in the simulation results.

   Another source of error in digital computer processing is related to the fact that only a fixed number of digits can be stored for any given number. This is termed round-off error. The matrix solving routine in WHAMO can be susceptible to inaccuracies due to round-off error. Contrary to truncation error, round-off error tends to increase with finer conduit segmentation and shorter time steps. Both the matrix solver and subsequent computations include accuracy checks, so that the user is warned when round-off error

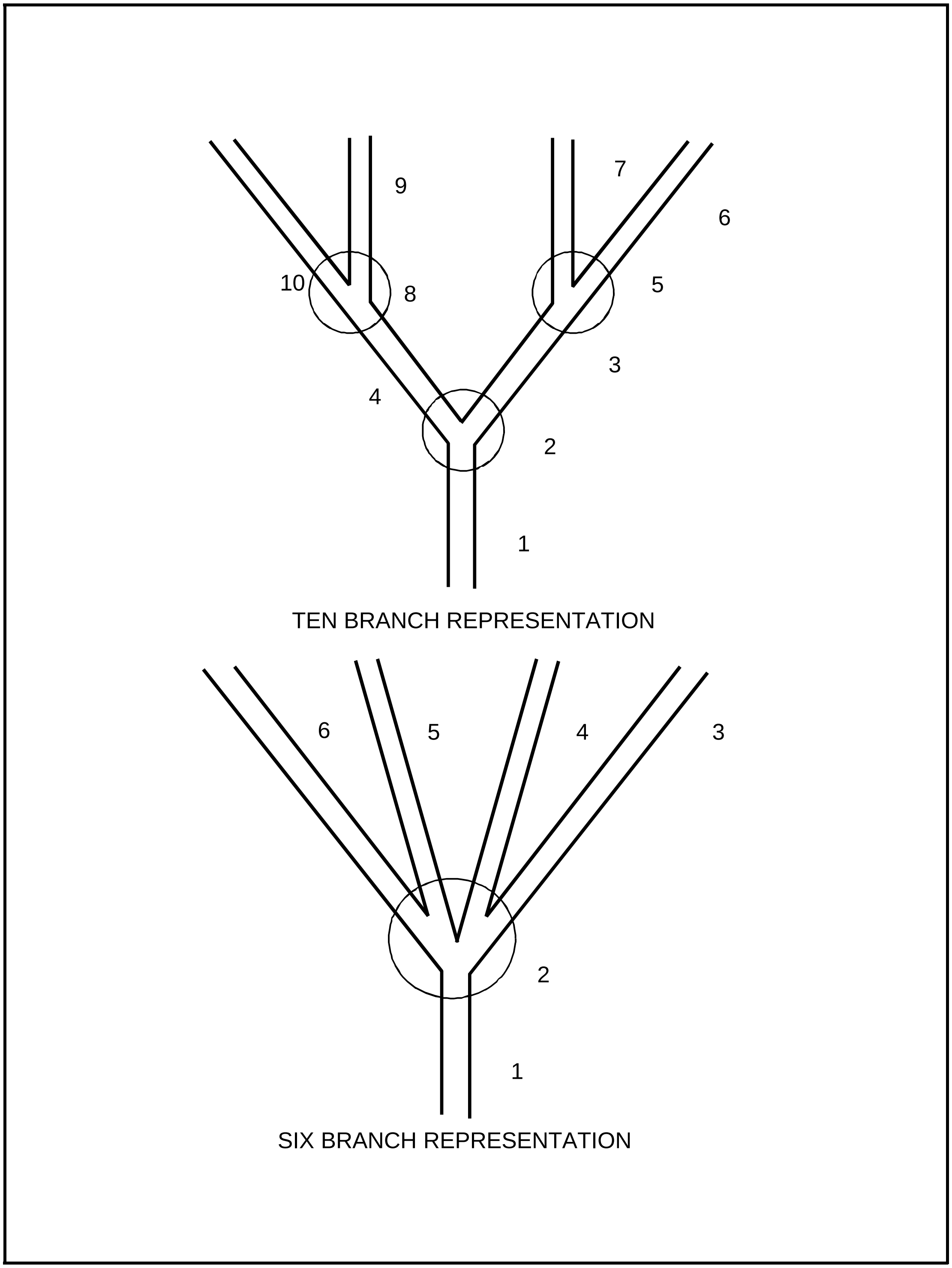


Figure 4. Two representations of four-way junction.

becomes significant. In this event, the number of segments for long conduits should be reduced and/or the time step length should be increased.

When the truncation errors are not random in nature but have a definite tendency to one direction or the other, the errors accumulate rather than compensate and large inaccuracies soon develop. This phenomenon is termed numerical instability. It was never encountered in the development and testing of the WHAMO model but has been a serious problem with other similar models. In those cases, numerical stability has been very sensitive to small adjustments in the computational weighting factor, THETA (Sections 4.4 and 6.4). In general, greater stability is achieved as the value is increased from 0.5 toward 1.0, but this cannot always be said to be the case. Generally, the default value of 0.6 gives good results. As mentioned in Section 2.1, when simulating very sharp transients due to pump shutdown or rapid valve closure, there may be some "noise" in the results. This "noise," a high frequency oscillation superimposed on the real solution in the vicinity of sharp wave fronts, can be minimized or eliminated by selecting a time step during the period of sharp transients which is an even multiple of the segment length divided by the celerity or, $\Delta t = n \Delta x/c$.

5. The results of any computer model should be reviewed very critically by the user. Besides making certain that the input data is accurate, he should experiment with different time step lengths and different segmentation of long conduits to be sure that the important features of a simulation are not sensitive to small changes in these computational parameters. He should also analyze the sensitivity of this simulation to changes in system and operational properties — such as friction factor, celerity, and gate closure rates — which may not be known precisely. The water hammer effect has often been found sensitive to very small changes in gate closure schedule. It is also recommended that a WHAMO user check turbo-machine simulation results to be sure that they adhere faithfully to the machine characteristics. This is done by using the applicable similarity relations presented in Chapter 6 to scale down the simulation output to be compatible with the characteristics, then plotting simulated operation along with the characteristics to be sure that they agree. Speed change can also be checked by making some hand calculations based on equation (6.24). Further, any other analytical tools which can reasonably be applied should be used to check simulation results. Finally, the user must test every simulation against his experience and understanding of the phenomena and be satisfied before accepting the computer model results.

3.3 Creating an Input File Using a Text Editor and Commands

3.3.1 Input Conventions

The WHAMO program employs a free format Problem Oriented Language (P.O.L.) input code. This is a command structure where each data value being entered is identified by an alphanumeric "tag" and thus assigned to its desired location. The Appendix contains reference indexes that may be used to locate input command instructions for particular aspects of WHAMO simulations. Input files are created with a word processor or text editor. Almost any of the commercially available word processing programs can be used. The only requirement is that the file must be saved in ASCII text format.

The input data consists of two levels of commands referred to as primary and secondary commands. The PRIMARY level commands are used to specify the type of data which follow, i.e., does the data refer to a conduit element, system connectivity, etc. Each primary command sequence will begin with a single alphanumeric word, contain a set of secondary commands, and terminate with a FINISH command. SECONDARY level commands normally will consist of an alphanumeric command tag followed by the required numeric data value or values. On occasion, a secondary command will consist solely of a single alphanumeric command.

In order to enhance user interaction with the system, the data will be read in a free format fashion. Using such a format, each command tag and/or data value will have no fixed location in an input line but will be simply separated from the other data by one or more blanks acting as delimiters of the data string. Such a format also permits the sequence of secondary commands within a primary command block to be as desired, rather than dictated by programming considerations. A typical command sequence in a data input file might look as follows:

CONDUIT	ID	Cl	DIAM	6	LENGTH	100.3	FINISH
	Tag	Data	Tag	Data	Tag	Data	
Primary Command	Secondary Command Block		Secondary Command Block		Secondary Command Block		Terminates Primary Command Block

Basically, the above commands serve to define a conduit element with the following properties:

Identifier:	Cl
Diameter:	6 (feet)
Length:	100.3 (feet)

Some other aspects of the input editor are worth noting:

1. Comments may be included in a set of secondary commands by enclosing them in parentheses (). The parentheses must be offset from both the preceding and succeeding data or comment by at least one space. All comment sequences should be terminated by a) or will be automatically terminated by the end of record.

2. Processing of an input record is terminated if an open bracket character [is read. The bracket must have at least one blank on each side of it.

3. Comment lines may be included between primary command blocks by use of the primary command, C.

C COMMENT IN PRIMARY SEQUENCE

4. It is possible to abbreviate all commands (primary and secondary) to four characters in length, e.g.:

COND and CONDUIT
DIAM and DIAMETER

are equivalent commands. It is recommended however, that enough of the command be retained to preserve the meaning of the command.

3.3.2 Command Groups and Input Sequence

Following is a list of the P.O.L. primary commands and a brief description of their use. These commands fall logically into five groups: those which define the individual system elements, those which define the interconnection of these elements within the system, those which specify the output information desired by the user, those which specify the operating conditions to be simulated, and those which initiate and control program execution.

Element Commands

The following primary commands are used to define the properties of the hydraulic elements. Each element in the network must be defined by such a command, followed by the data which specifies its hydraulic properties.

CONDUIT	Defines a circular, closed conveyance element which has constant properties along its length (except that the diameter may vary gradually).
DCHANGE	Defines an expansion or contraction element between conduits of different diameter.
FLOWBC	Defines a specified discharge condition at a boundary of the system.
GOVERNOR	Defines a governing mechanism for pump-turbines and turbines.
ONEWAY	Defines a non-return or check valve.
PCVALVE	Defines a pressure control valve.
PUMP	Defines a pump.
P-T	Defines a machine which can be operated in either the pump or turbine mode.
RELIEF	Defines a pressure actuated valve. The function of this element is similar to the PCVALVE element, but operation is specified in a different manner.
RESERVOIR	Defines a free water surface boundary which may be headwater, tailwater, or a canal, etc.
SURGETANK	Defines a simple surge tank, one way surge tank, Johnson differential surge tank, or air chamber surge tank.
TJUNCTION	Defines a three way Tee Junction for which head losses are calculated across the junction.
TURBINE	Defines a turbine.
VALVE	Defines a throttling device of a certain type.

Machine and Valve Characteristics Commands

These commands provide further definition of machine and valve elements. Each set of characteristics may apply to any number of machine and valve elements.

PCHAR Defines the characteristics of each type of pump with dimensionless four quadrant curve data of head and torque versus speed and discharge.

PTCHAR Defines the characteristics of each type of pump-turbine with model data relating discharge and torque to speed and wicket gate opening.

TCHAR Defines the characteristics of each type of turbine with prototype or model data relating head (or phi), wicket gate opening, power, and discharge.

VCHAR Defines the characteristics of each specific type of valve with a discharge or head loss coefficient versus opening curve.

System Command

The following primary command is used to specify the interconnection of all the system elements except the GOVERNOR element (also, VCHAR, PCHAR, TCHAR, and PTCHAR, which are used to define the characteristics of certain elements and not the elements themselves).

SYSTEM This command specifies the connection of the system elements by defining the node points which bound each element.

Output Commands

With these primary commands the user can specify the form and extent of the computer output.

DISPLAY Specifies which portions of the input data should be displayed in tabular form as part of the printed output.

ECHO/NOECHO Specifies whether the input commands should be echoed in the output or not. Input commands are echoed by default.

HISTORY — Specifies the elements and variables for which time histories are to be printed as part of the computer output.

PLOTFILE — Specifies the elements and variable for which time histories are to be saved for plotting by WHAMGR.

SNAPSHOT — Specifies printing of the heads and discharges throughout the system at certain times.

SPREADSHEET — Specifies the elements and variables for which time histories are to be saved to file in a spreadsheet compatible format.

TEXT — Used to enter a text message to be printed with the header of simulation results output tables.

Simulation Commands

With these primary commands the user controls the system operating conditions which are to be simulated, and the computational parameters of the simulation.

CLOSURE — Defines a dimensionless, non-linear curve for the "cushioning" portion of the special gate closure option.

CONTROL — Controls the computation parameters such as the time step and the numerical weighting ratio.

FLUID — Specifies density for fluids other than water.

INITIAL — Allows specification of preliminary estimates of discharge, head, and/or wicket gate opening to be used in the initial conditions determination.

OPPUMP — Specifies the operation of each pump.

OPPT — Specifies the operation of each pump-turbine.

OPTURB — Specifies the operation of each turbine.

SCHEDULE — Defines the operating schedule for valves, turbine gates, and generator loads, as well as the head or flow schedules for boundary conditions.

SEPARATION Specifies the pressure below which a message will be printed warning of possible water column separation.

Execution Commands

A number of commands control the execution of the simulation run. These include the CHECK, GO, IONLY, NEWRUN, RERUN, and GOODBYE commands.

Input Sequence

Except for execution commands, WHAMO primary commands may be input in any sequence. A recommended input sequence of command groups, corresponding with the data preparation steps given in Section 3.2, is presented below:

1. Simulation Title (must be first line)
2. SYSTEM Command
3. Element Commands
4. Machine and Valve Characteristics Commands
5. Output Commands
6. Simulation Commands
7. IONLY or CHECK (if applicable)
8. GO (indicates end of input data for this simulation)
9. GOODBYE (must be last line)

3.4 Creating an Input File Using the WHAMO Network Builder

WHAMO Network Builder is a Windows application which is used to graphically build the system network and create the WHAMO input file. WHAMO and Network Builder are linked by the WHAMO input file. Through Network Builder, you can construct the system network, input the element descriptions and operational information, and provide the simulation control information. Network Builder exports a WHAMO input file which contains all the information necessary to run a WHAMO simulation.

IMPORTANT NOTE: There is a difference in network construction philosophy between using the Network Builder and using a text file to define the network. In Network Builder, two system elements which are not conduits cannot be joined together directly. Conduits must be used to link these elements. This is different than the text file method, which does allow elements that are not conduits to be joined together directly.

3.4.1 Starting WHAMO Network Builder

The following instructions assume that Network Builder is already installed on your computer. If it is not, see Section 3.1 for installation instructions.

Start WHAMO Network Builder by clicking the Windows 95 “Start” button and selecting “Run”. Click the “Browse” button and go to the directory in which the WHAMO Network Builder was installed. Select BUILDER.EXE, then click OK and the program will start. You may also start BUILDER.EXE from the Windows Explorer. Frequent users may find it convenient to create a shortcut to BUILDER.EXE and place it on the Windows 95 desktop.

The Network Builder main menu window will appear on the screen (Figure 5). The menu bar contains eight selections that control the basic operation of Network Builder. The eight selections on the menu bar are:

File Controls program functions such as file opening and closing plotting, printing, and exiting

Edit Provides input windows for computational parameters, machine component characteristics, and schedules; and controls for some screen manipulations

View Controls the display characteristics of the Main Menu Window

Structure Controls the display grouping of system elements

Align Controls the alignment of system elements

Graphics Controls graphics options

Window Toggles display window

Help Online help facility

In addition to the menu bar, an input grid and icon palette will appear. The icon palette contains icons which represent the different network elements, as well as graphics/text features which can be added to enhance the network diagram.

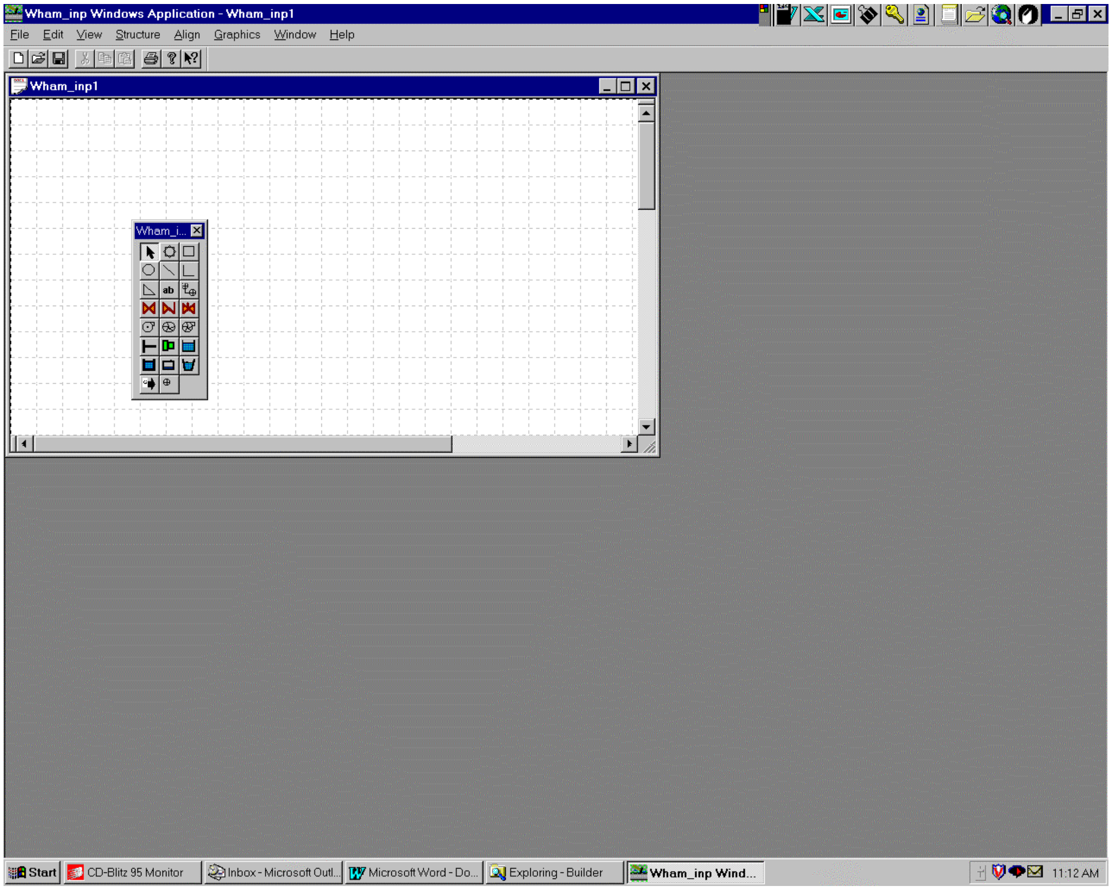


Figure 5. WHAMO Network Builder opening screen.

3.4.2 Creating a WHAMO Network

A blank input grid is displayed when Network Builder is started (see Figure 5). Additional blank input grids can be obtained by clicking on "File" on the main menu bar, then on "New". Whenever a new input grid is displayed, an icon palette is also displayed.

The network schematic is created by placing system elements (valves, machines, boundary conditions) on the input grid, then linking them with conduits. The program prompts for descriptive information about each component after it is placed on the grid. Detailed instructions for placing and describing system components are given in the following sections.

3.4.2.1 Elements. The first step in creating a WHAMO network is to define and place the system elements. These elements include valves, turbines, surge tanks, and boundary elements. All elements are added by clicking on the appropriate icon on the icon palette, then moving the cursor to the desired location on the grid and clicking again. Hold the cursor over an icon to display its name or function. The required number of nodes will be automatically associated with the element (for example, two nodes for a valve: an inflow and an outflow node; and one node for a surge tank). The position of these nodes can be changed to simplify the layout of the system. If you click on the node, a dashed box will appear. Move that box to the desired location of the node.

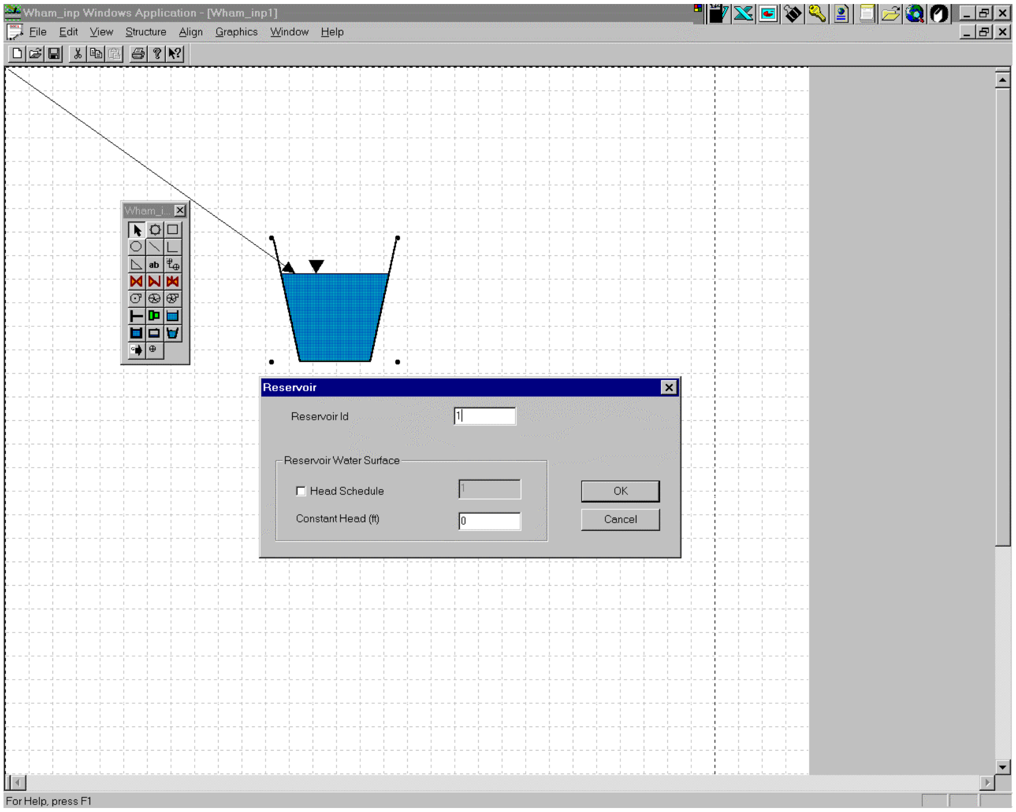


Figure 6. Reservoir data entry screen.

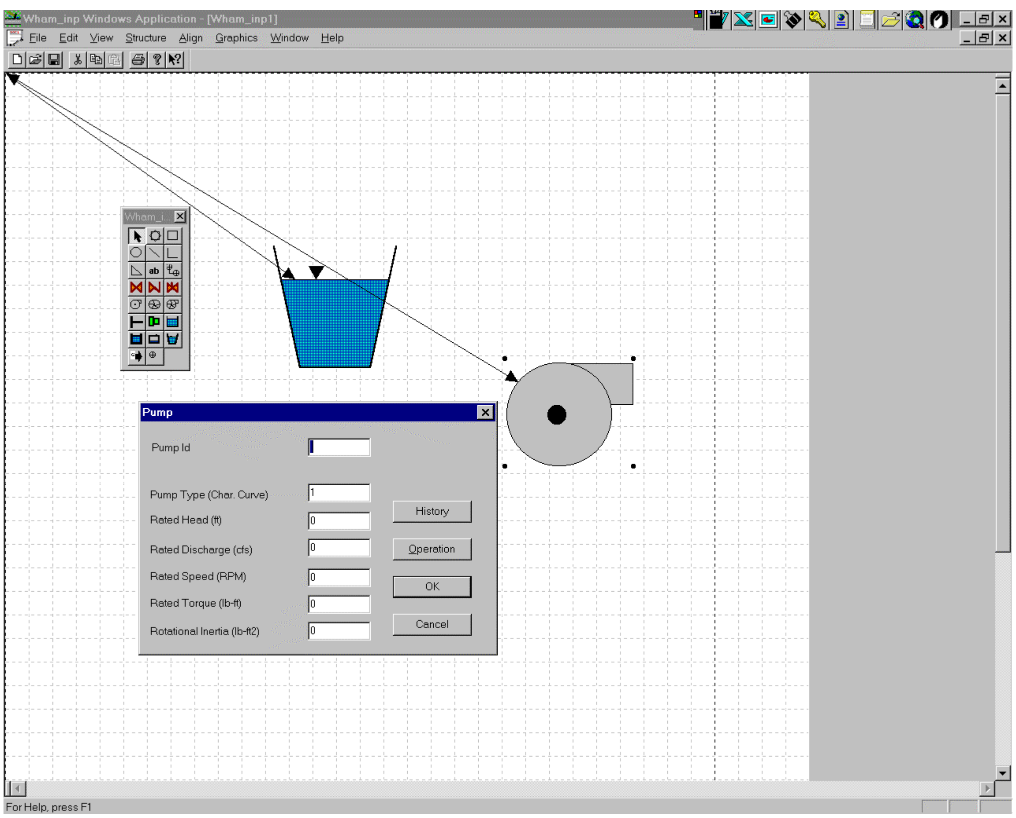


Figure 7. Pump data entry screen.

After you have placed each element, you will be prompted for data that further describes it. Figure 6 shows a reservoir that has been placed on the grid and expanded in size, along with its data entry box. Figure 7 shows a pump and its data entry box. It is usually easiest to enter the descriptive data as the network is created, however, it may also be added or edited later on by simply clicking on the element. The appropriate dialog box will appear and you may enter or edit the data. Table 2 lists the available elements and describes the data requirements for each. For further details on any of the data requirements, consult the Command Reference in Chapter 4.

Table 2. Data requirements for system elements.

Element Category	Element Name	Descriptive Data Requirements
Flow Elements	*Diameter Change Element*	A diameter change element is used to represent head losses for locations where there is a significant change in conduit diameter. The positive and negative head loss coefficients can be specified directly, or you can specify the geometry of the diameter change, and the head losses will be computed.
	Control Valve	Within the valve input dialog box, you are asked to identify the valve type and assign a valve schedule number to it. If the valve type is undefined, then you must input the valve characteristics through the **Edit** option of the Main Menu Bar as discussed in Section 3.4.3. The valve schedule is also entered through the **Edit** option.
	One Way Valve	The physical parameters of the one way valve are entered through the input dialog box. You can reverse the direction of allowed flow through the input dialog box.
	Pressure Control Valve	The physical parameters and flow characteristics are entered through the input dialog boxes. If the valve is user defined, then you must input the valve characteristics as discussed in Section 3.4.3.
Turbo Machines (cont'd on next page)	*Pump*	The required physical parameters of the pump are entered through the pump input dialog box. If the pump has the same physical parameters as a previously defined pump then the parameters can be transferred as described for conduits, and re-specification of the parameters is not needed. You may assign the pump a characteristic type through the pump input dialog box. The pump characteristic data is entered through the **Edit** option of the Main Menu Bar. The operation specification of the pump is also entered through the pump input dialog box.

Turbo Machines (cont'd)	*Turbine*	The required physical parameters of the turbine are entered through the turbine input dialog box. If the turbine has the same physical parameters as a previously defined turbine then the parameters can be transferred as described for conduits, and re-specification of the parameters is not needed. Operation parameters are also transferred in this process. The turbine characteristics are entered through the **Edit** option of the Main Menu Bar. The gate schedule is entered as a valve schedule through the **Edit** option, and a generator load schedule is entered as a load schedule through the **Edit** option. The operation specification of the turbine is entered through the turbine input dialog box. An estimate of initial gate position and head is needed only if you are simulating governor control.
	Pump-Turbine	The physical parameters and schedules for the pump-turbine are entered in the same manner as for the pump and turbine described above.
Boundary Elements	*Flow Boundary*	When you add a flow boundary condition, an input menu for the boundary element appears. The boundary ID must be specified together with either a constant discharge amount or a discharge schedule. The discharge schedule is specified by checking the flow discharge box and assigning a schedule number in the accompanying box. The actual discharge schedule is entered through the schedule menu (discussed in Section 3.4.3).
	Free Surface Boundary	A free surface boundary is created by inserting a reservoir element. You must specify the reservoir ID, and the reservoir water surface elevation as either a constant elevation or an elevation schedule. If specifying an elevation schedule the head schedule box is checked, and a schedule number is assigned. The actual head schedule is entered through the schedule menu (discussed in Section 3.4.3).
	Simple Surge Tank	Through the input dialog box, enter the surge tank ID and the physical parameters of the surge tank as specified in Chapter 4.
	Differential Surge Tank	A Johnson differential surge tank can be represented by a differential surge tank element. Specify the differential surge tank ID and the physical parameters of the differential surge tank as specified in Chapter 4.
	Air Chamber Surge Tank	For an air chamber surge tank, you must specify the element ID, and physical and thermodynamic properties of the surge tank.

3.4.2.2 Conduits and Flow Direction. In Network Builder, conduits are used to link elements, and no nodes are automatically associated with a conduit. The conduits can represent actual physical pipes, or can serve as a dummy link between two elements. The start and end point elements of a conduit must be placed on the grid (as described in 3.4.2.1) before the conduit can be added. Those elements are then linked by clicking on the link (conduit) icon, and then clicking on the nodes associated with the start and end point elements.

It is important to maintain a consistent direction of flow throughout the system network. When elements such as pumps or valves are added to the system schematic, a default direction of flow is indicated through the associated nodes. This direction can be reversed by moving the position of the associated nodes from one side of the element to the other. In a conduit the positive direction of flow is in the direction of the end point of the conduit.

The required descriptive information for conduits is entered through the element dialog box. Entrance and exit losses, and additional losses can also be specified. Conduits with variable diameters can be specified by checking off the variable diameter box, and then specifying the length and diameter of each section through the variable diameter schedule box. If multiple conduits have exactly the same physical properties, after one conduit is specified its properties can be transferred to other conduits. To do this, left click on the conduit to be copied, select Copy from the Edit Menu on the Main Menu Bar, then click on the new conduit with the right mouse button. If properties have been transferred then a message appears on the screen which says "Conduit properties transferred." A conduit can also be designated as a dummy conduit by checking off the dummy conduit box, in which case no descriptive information is required.

3.4.2.3 Nodes, Simple Junctions. As previously mentioned, nodes are automatically associated with system elements. Additional nodes can be added to the system as needed, for example when joining two or more conduits. A simple junction is therefore created by adding a new node to the system and connecting 3 or more conduits to that node. No descriptive information about a node is required. You can specify the elevation of the node and request time history data of discharge and head at the node.

3.4.2.4 Removing Elements or Links, Revising Connectivity. You can revise the network and can remove elements or conduits from the network. To remove a system component, select it, then click on Delete under Edit on the Main Menu Bar. It is very important to remove components with the Delete option instead of the Cut option, because Cut may not remove the component completely from the network. You must remove both the element and the associated nodes when removing an element such as a pump from a network. If you delete either the start point node or end point node of a conduit, then the linking conduit will also be deleted.

3.4.2.5 Tee Junctions. Tee junction elements are used in place of simple junctions in order for WHAMO to automatically compute head losses at junctions of 3 conduits. Only conduits or dummy conduits can be connected directly to tee junctions. The tee junction parameters are defined in Chapter 4.

3.4.3 Inputting Schedules and Characteristics

Schedules and machine characteristics are input through the Edit option on the Main Menu Bar. To enter machine characteristics, select Characteristics, then the class of machine to be described: Valve, Pump, Turbine, or Pump-Turbine. For any of these elements, enter the assigned type number (corresponding to the assigned type number from the element dialog box), and then enter the characteristics, as described in Chapter 4, through the characteristics dialog box. Figure 8 shows the pump characteristics dialog box.

To enter a schedule, select Schedule and then the type of schedule to be entered: Valve, Load, Head Boundary Condition, or Flow Boundary Condition. Once the schedule dialog box is open, click Add Schedule, then enter the schedule data. The schedule information is entered as described in Chapter 4. Figure 9 shows the valve schedule box.

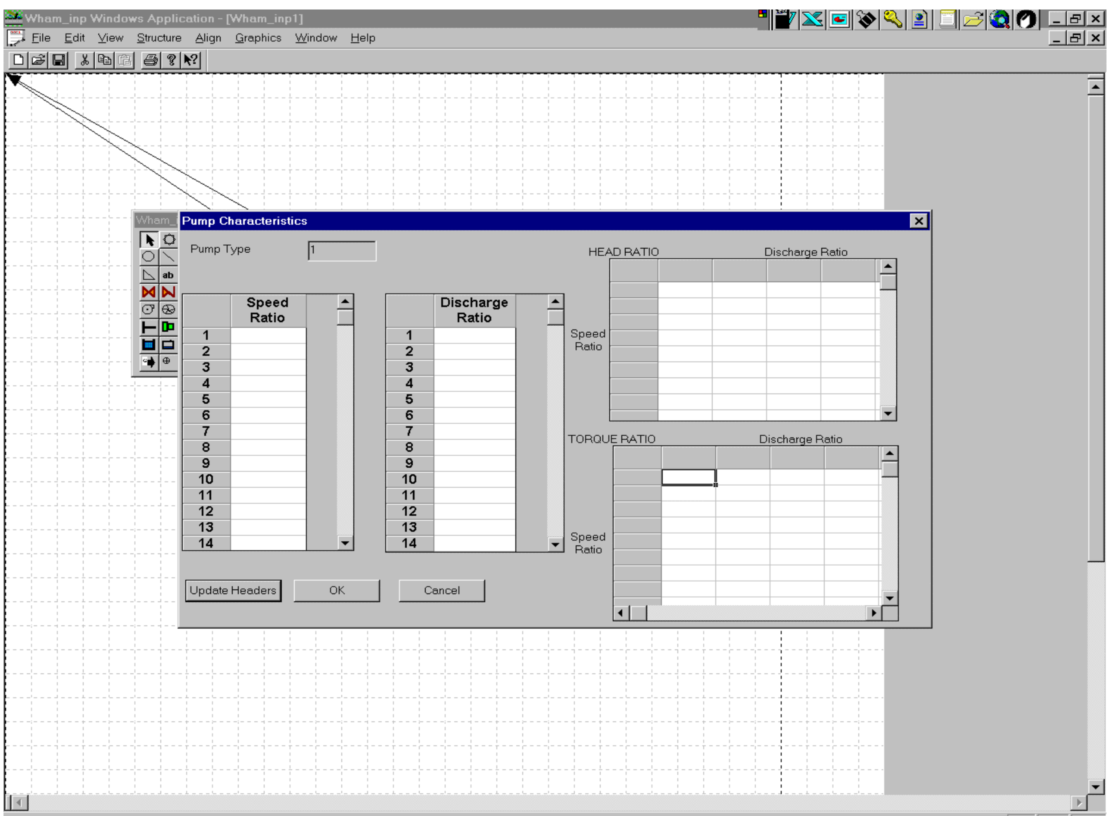


Figure 8. Pump characteristics box.

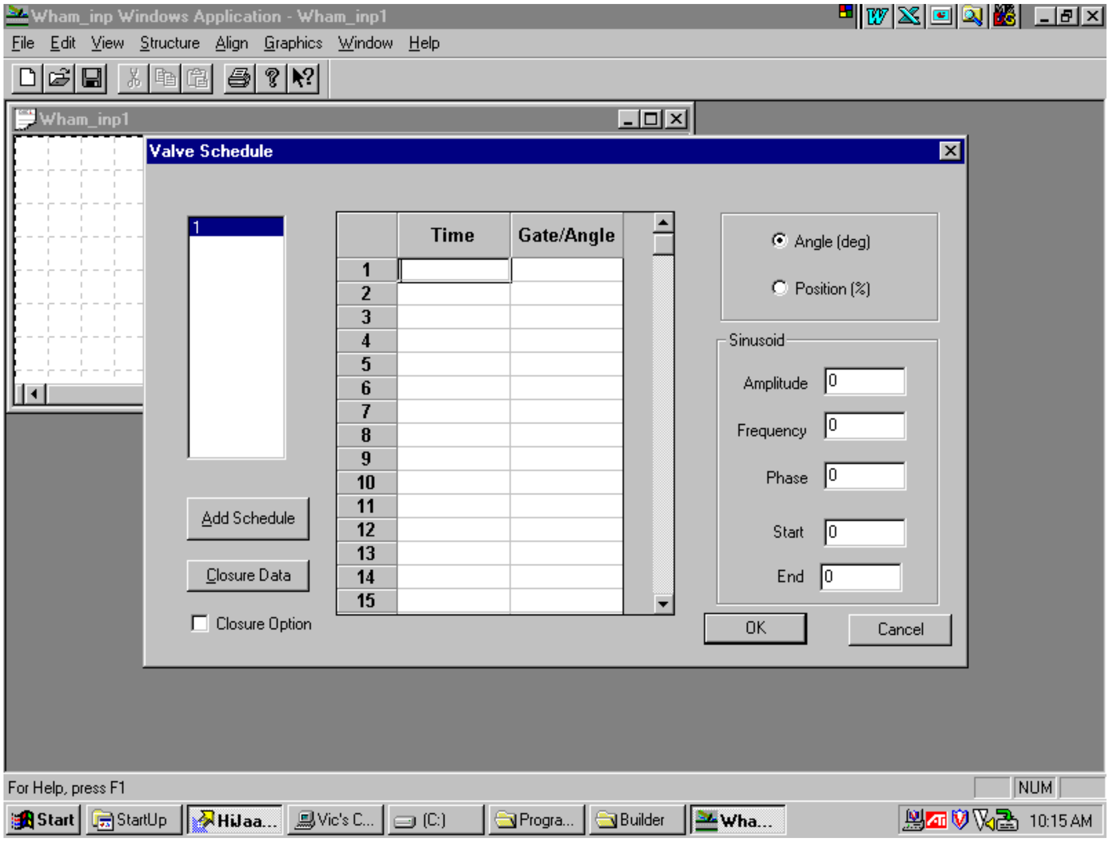


Figure 9. Valve schedule dialog box.

3.4.4 Output Control

Element specific time history data is specified through the input dialog box for that element. Figure 10 shows the time history box for the pump that was shown in the previous figures. Click on the history box , then choose one or more of the output parameters to be displayed. Choose the file type to which the output is sent. Output data can be written to a simulation log file, a plot file to be plotted in WHAMGR, or a spreadsheet file. Some element types provide multiple history buttons. This provides flexibility to send the various parameter data to different output files, or to send the data to multiple output files. Time history data for flow boundary and free surface boundary elements can be obtained by specifying time history output for the nodes associated with those elements.

The formatting of output is controlled through the Edit option on the Main Menu. Select Output Options from the options under Edit, then select History Format and specify the format you wish to use.

You can also specify what input data is printed out in tables. Select Edit from the main menu bar, then Output Options, then Display. Select which input parameters are to be repeated in the output (Figure 11).

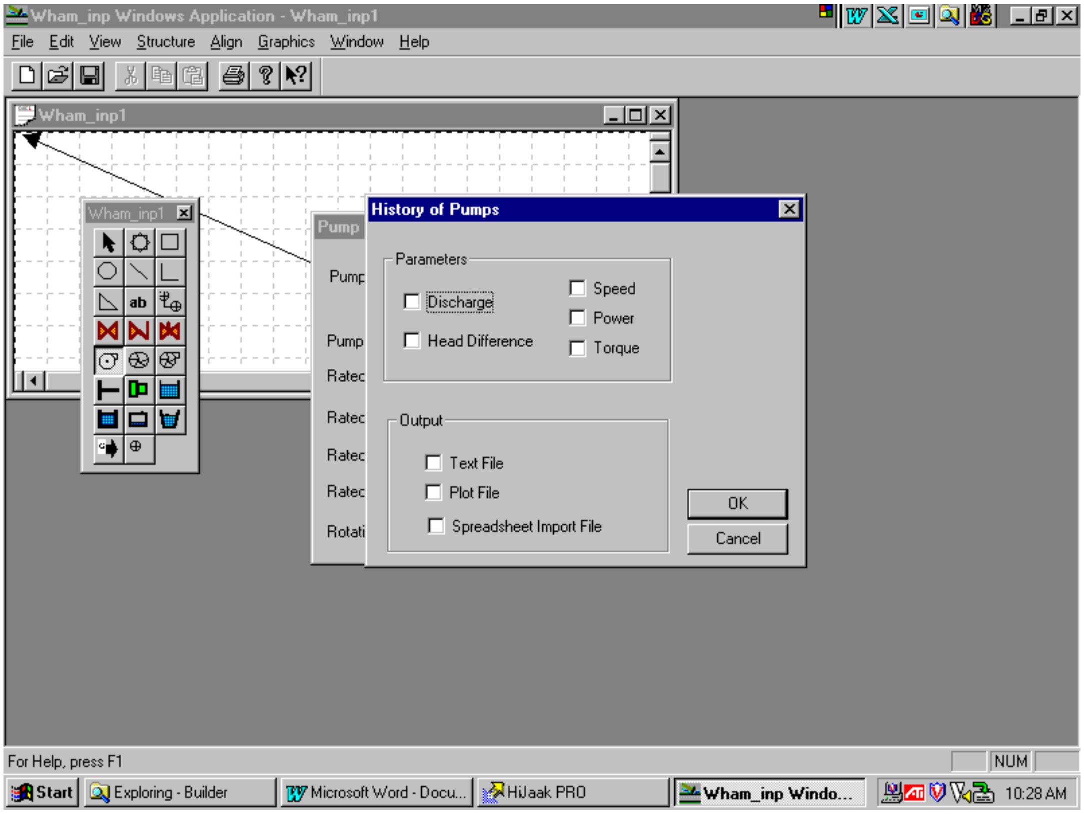


Figure 10. Time history dialog box.

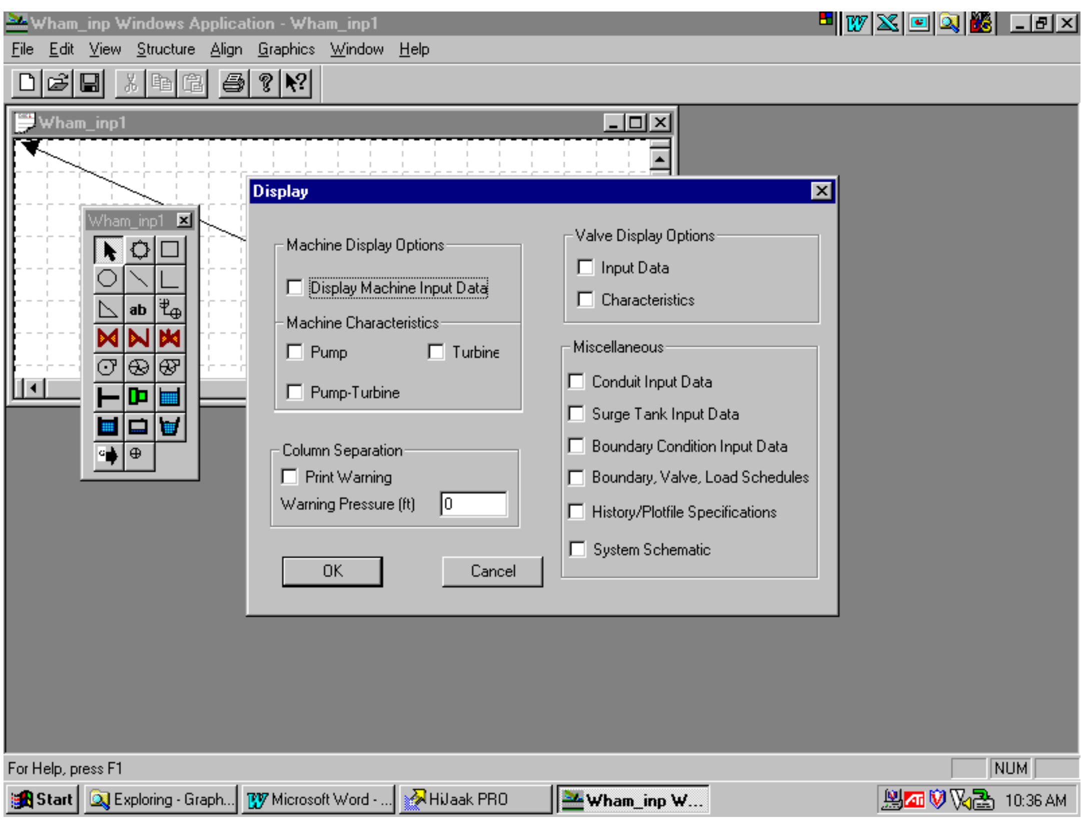


Figure 11. Display options dialog box.

3.4.5 Simulation Controls

The execution controls for a simulation are entered through the Edit option on the Main Menu Bar. Select Control from the options under Edit, and then enter the simulation control parameters as described in Chapter 4.

3.4.6 Specifying Initial Branch Estimates

The specification of initial branch discharge is an infrequently used function, and is not normally required input for a WHAMO simulation. However, if the steady state computation does not converge to a solution, it may be necessary to input estimates of initial conditions. To specify these conditions, input the serial branch number and the estimated initial steady state discharge by selecting Initial Branch Flow under the Edit option on the Main Menu Bar. A previous simulation of the system is required to obtain the serial branch numbers from a printout of the system schematic in the log file.

3.4.7 Saving and Retrieving WHAMO Schematic Files

Network Builder saves the WHAMO network, element characteristics and schedules, and simulation control information to a *.whm file type. This file can be subsequently read by Network Builder to revise the model network. This file type is not readable with a text editor or by WHAMO.

Clicking on the File menu item displays 12 sub-menu items: **New, Open, Close, Save, Save As, Export, Print, Print Preview, Page Setup, Print Setup, Recent Files**, and **Exit**.

WHAMO schematic files can be saved using either the Save or Save As commands. When saving a newly created schematic file for the first time the Save and Save As commands function in the same manner. A dialog box appears which allows you to specify the name of the file, the file type (either *.whm WHAMO schematic, or *.* all files as yet undefined), and the directory path to where the file will be saved. When saving edits to a file that had previously been saved, the Save command saves to the default schematic file name.

When Network Builder is started the most recently edited schematic files are listed in the sub-menu, and can be retrieved by clicking on the desired file name. Otherwise, clicking on Open creates a dialog box for listing directories and files from which to select a data file for retrieval. Click on the desired schematic file (causing it to be highlighted) then click Open to retrieve the file.

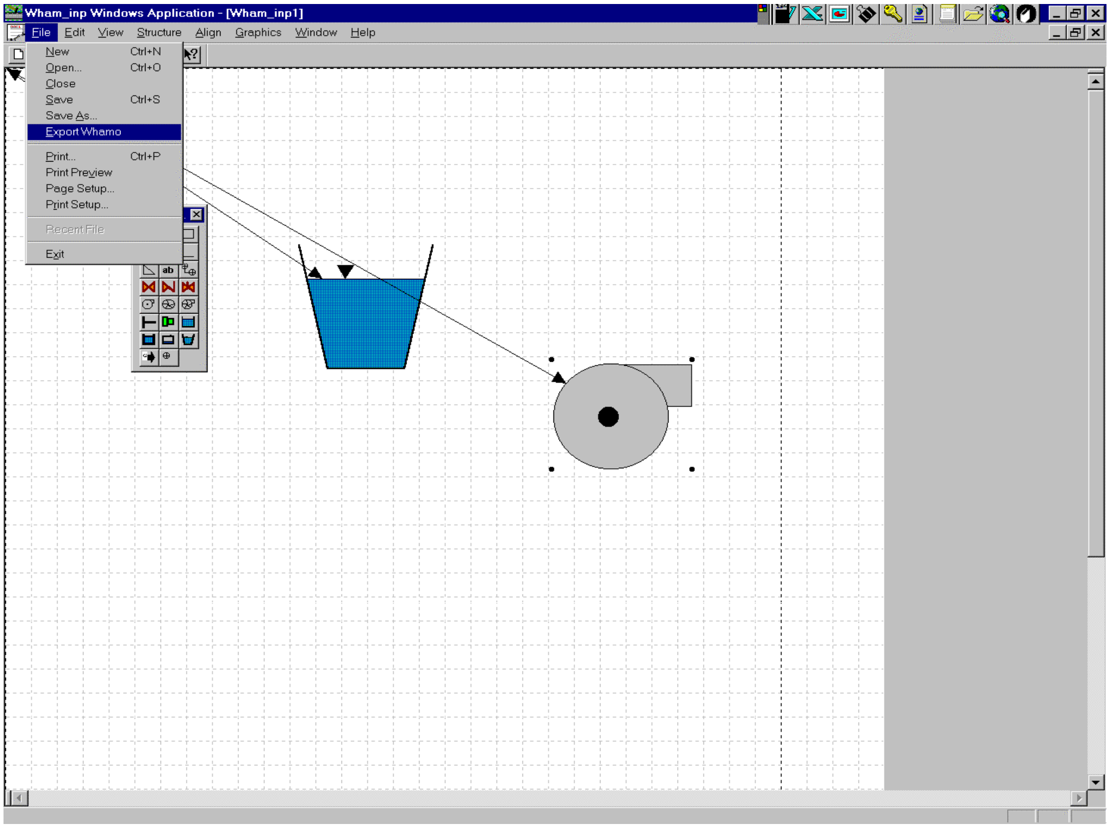


Figure 12. Exporting a file to WHAMO.

3.4.8 Exporting to a WHAMO Input File

Creating a WHAMO input file is accomplished using the Export command on the File menu (Figure 12). You must do this for WHAMO.EXE to be able to run the simulation. Only files with the *.inp file extension are created using Export. The WHAMO input file contains all the information required to run a WHAMO simulation. The WHAMO input file is an ASCII file which can be edited with a text file. Input files cannot be reopened and edited in Network Builder.

3.4.9 Printing

The diagram of the WHAMO schematic can be printed. Clicking on the File menu item will display the following submenu items related to printing:

Page Setup You can set the orientation of the figure on the page, either portrait or landscape, and can set the width of the borders

Printer Setup You can select the printer destination, paper selection, and other printer settings

Print Preview Allows you to preview the schematic as it will appear on the printed page.

3.4.10 Modifying View

There are several options for modifying the appearance of the input screen. These options are found under the View item on the Main Menu Bar.

Grid When the Network Builder opens an input screen, a grid automatically appears. You can remove the grid by removing the check next to **Grid Visible** under the **View** option. When components are added to the network, by default they snap to the nearest grid square corner. You may turn off this option by removing the check next to **Snap to Grid**. This allows flexibility in locating network components.

Zoom Controls the size of the entire network image. Click on the option **Zoom**, then either choose a pre-set magnification or define a specific magnification. You can also interactively re-size individual network components by clicking on the component and re-sizing the rectangular box that pops up associated with that component.

Display Element IDs Toggles the display of element IDs on and off. Select **View** from the Main Menu Bar, then click on **Display Element IDs** to toggle the display.

Display Node IDs Toggles the display of node IDs on and off. Select **View** from the Main Menu Bar, then click on **Display Node IDs** to toggle the display.

Aligning Network Components Network components are aligned using the options found under **Align** on the Main Menu Bar. Select the components to be aligned, then choose the type of alignment you wish to apply from the **Align** menu.

3.4.11 Enhancing the Diagram

Several options are available to you if you want to enhance the network schematic diagram. You can add text, lines, and shapes to the diagram without altering the system network. These are added by clicking on the appropriate icon on the palette and then placing the shape or text at the desired location on the diagram. When you select "text," you can type the desired text into the box that appears.

Through the Graphics option on the Main Menu Bar, you can customize the color, font, and fill of lines and text added to the diagram. Changes to the color and font settings impact only those shapes or text subsequently added to the diagram. Previously added text or shapes are unaffected. You can also import bitmaps into the diagram using the Import Bitmap option found under the Graphics menu item.

3.4.12 Online Help

Online help is available in Network Builder. Select Help from the Main Menu Bar, then select Help Topics. Choose a topic from the contents page that appears.

3.5 Running the WHAMO Simulation

WHAMO is executed under the Windows 95 operating system. This manual assumes you are familiar with the Windows 95 environment including the use of a mouse and window controls; if not, please refer to your Windows 95 documentation.

To run WHAMO, press the Windows 95 “Start” button and select “Run.” Click the “Browse” button and go to the directory in which WHAMO was installed. Select WHAMO.EXE, then click OK and the program will start. Or, you may simply double-click on the file WHAMO.EXE. You may also start WHAMO.EXE by double-clicking on it in the Windows Explorer. Frequent users may find it convenient to create a shortcut to WHAMO.EXE and place it on the Windows 95 desktop.

WHAMO runs in an MS-DOS text window. When the program starts, you will be prompted to type the names of 4 files associated with the simulation you are running. They are:

Input Data File This is the ASCII text file which contains the input commands and data used to define a WHAMO simulation. The file is created using a text editor or word processor as described in Section 3.3, or by using the WHAMO Network Builder as described in Section 3.4.

Output File This is an ASCII text file that will be created by WHAMO and will contain tabular input data and results for this simulation.

Press <enter> to create a file with the same root name as the input file with an extension of .OUT. To give the file a different name, type it at the prompt. However, using the default name makes it easier to keep track of your work.

Plot File This is an unformatted file that will be created by WHAMO for storing simulation results for graphical plotting by the WHAMGR program. Type <enter> to create a file with the same root name as the input file with an extension of .PLT. To give the file a different name, type it at the prompt. However, using the default name makes it easier to keep track of your work.

Spreadsheet File This is an ASCII text file created by WHAMO for storing simulation results in a tabular form generally compatible with spreadsheet programs. Type <enter> to create a file with the same root name as the input file with an extension of .TAB. To give the file a different name, type it at the prompt. However, using the default name makes it easier to keep track of your work.

After the file names have been input, the WHAMO simulation will run without further user interaction according to the instructions contained in the input file. The current simulation time will be displayed on the screen so that you can track its progress. Diagnostic and error messages generated by the WHAMO program, if any, will be written to the specified output file. When the simulation is finished, the DOS window in which WHAMO was running will close. You are now ready to view the results.

3.6 Diagnostics

The WHAMO model provides extensive error checking for input syntax, and for completeness and reasonableness of the input data before embarking on the simulation computations. A diagnostic message is printed for each error when it is detected. These messages are self-explanatory. As with any system of error diagnosis, a single input error can lead to numerous diagnostic messages as the effect of that error is felt in different parts of the program. Particular attention should be paid to the first diagnostic message. One of the most common input errors is to omit the FINISH statement at the end of a primary command group. When this occurs, a diagnostic will be printed indicating that the subsequent primary command is not a recognized secondary command under the previous primary command.

If the steady state computation fails to converge, a message is printed which directs the user to assist the program by including an INITIAL command with the input. This message includes a display showing the program designated branch numbers. If steady state convergence cannot be achieved, even with the use of the INITIAL command, it may be necessary to begin the simulation with no flow by closing a valve or wicket gates, then slowly bring the system to the desired initial state during the transient simulation.

As mentioned previously, the program checks during the computation for inaccuracies resulting from round-off error. On detection of such an error, a message is printed which begins either

*** LETAM-TEST ***
or
*** SOLVER ***

If one of these messages is encountered, the user should adjust the segmentation of conduits or the time step as described in Section 3.2. If ONEWAY or RELIEF elements are used, a minimum time step length can be specified with DTMIN under CONTROL.

Other errors encountered in the computation phase of the program will cause the computation to terminate, and the results up to that time will be printed followed by a "fatal" error message. Two examples of this type of error message are as follows:

*** TRANS *** NO CORRECT TRANSIENT SOLUTION
AFTER____ ITERATIONS AT TIME____,-- FATAL

*** SETPT *** AT TIME__ , NO CONVERGENCE TO
COEFFICIENTS OF LINEAR H-Q EQUATION
FOR ELEMENT___ -- FATAL

If these or other errors are encountered during computation, the cause could be an obscure error in the input data. In some cases, reducing the length of the computational time step is the solution. Another likely source of problems is the machine characteristics data. These data need to be carefully checked for transcribing errors. Moreover, if these data are not "smooth," if not enough table points have been supplied to insure precise interpolation, or if the simulated conditions exceed the bounds of the characteristic tables, program computations may not converge. The characteristics plotting capability of WHAMGR should be utilized to check for input errors or poor interpolation. In general, if computational time steps of about 0.025 seconds are not short enough to get past a "sticky" point in the simulation then the characteristics data should be checked and augmented in the region where difficulties occur.

As mentioned above, if the simulated operating condition of a machine exceeds the range of the characteristics table, computational difficulties may arise. Warning messages will be printed, for example:

NXSTEP SIMULATION MAY EXCEED RANGE OF
CHARACTERISTICS FOR ELEMENT____AT TIME___-- WARNING

The user must take particular care that sufficient characteristics data are input, especially for turbine simulations where machine speed will change significantly (whether model or prototype scale data are given).

3.7 Using WHAMGR to Display WHAMO Results

As explained previously, WHAMGR works in conjunction with WHAMO to plot simulated time histories and machine characteristics. WHAMGR is a Windows 95 application and can be installed and run on any Windows 95-compliant computer. Likewise, WHAMGR can be used with any true Windows-compiant monitor and printer.

WHAMGR and WHAMO are linked by two types of data files. The first type is the WHAMO input file that contains characteristics of WHAMO machine components: valves, pumps, turbines, and pump-turbines. Creation of WHAMO input files is described previously in this manual. The second type of file WHAMGR reads is a plotting file produced by WHAMO during simulation. Such files contain simulated time histories of variables such as discharge, head loss, and piezometric head at nodes and elements specified in the simulation input.

3.7.1 Running WHAMGR

WHAMGR is run from the Windows File Manager which is located in the Main window. To run WHAMGR, start the File Manager and go to the directory in which

WHAMGR was installed (as discussed in the Section 3.1). WHAMGR.EXE should be listed by the file manager.

To start WHAMGR, simply double-click on the file WHAMGR.EXE. This will result in the WHAMGR Main Menu Window appearing on-screen. The menu bar contains three selections that control basic operation of WHAMGR:

File controls program functions such as file opening and closing, plotting, printing, and exiting
View toggles Windows tool displays
Help a help facility (not currently implemented)

These selections are discussed in the subsections that follow.

3.7.2 Retrieving Data Files

Clicking on the **File** menu item causes four sub-menu items to be displayed: **Open, Print Setup, Files,** and **Exit.**

If the desired data file is listed in the sub-menu, then it is retrieved by clicking on that file name. Otherwise, clicking on **Open** creates a dialog box for listing directories and files from which to select a data file for retrieval. Click on the desired directory file (causing it to be highlighted) then click OK to retrieve the file. The **List Files of Type** button at the bottom of the dialog box is designed to list files with a .PLT extension, typical of time history data files, or an .INP extension for plotting machine characteristics from a WHAMO input file. Files with extension (*.*) may also be listed.

An empty WHAMGR plot window will appear after you have selected a time history data file or a WHAMO input file. The window will contain the message “No Selection.” PLEASE NOTE: “NO SELECTION” IS NOT AN ERROR MESSAGE. It simply means that you have not yet selected which parameters you wish to display in the WHAMGR plot window.

You will also notice that the main menu bar selections have changed. If you have selected a WHAMO time history plot file, the following menu bar items are displayed: **File, Data..., Options, View, Window,** and **Help.** These can be seen near the top of Figure 13. If you have selected a WHAMO input file, the menu display is almost the same: **File, Data Select, Options, View, Window,** and **Help.**

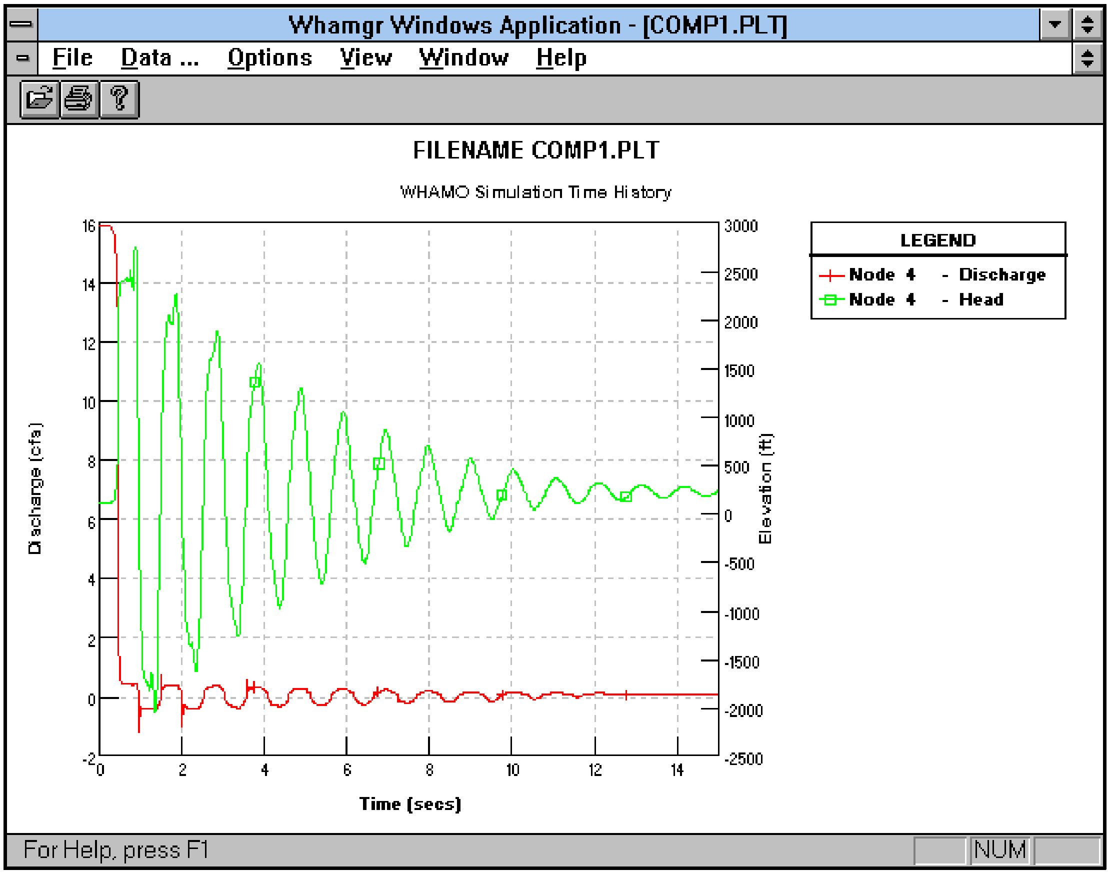


Figure 13. WHAMGR display of plot file.

The menu items **File, View, Window,** and **Help** are identical for both types of files. **File** is discussed in the present section and Section 3.7.5. **View, Window,** and **Help** are described in Section 3.7.6. The functions of the menu items **Data...**, **Data Setup,** and **Options** depend on the type of file loaded. These functions are discussed in Section 3.7.4.

3.7.3 Making Plots

Time Histories

Clicking on the **Data...** menu item creates a dialog box containing a list of elements and nodes and associated variables for which simulation results can be plotted. Element/node and variable combinations can be selected by clicking on the combination desired. More than one element/node and variable combination can be plotted. Accept the selections by clicking on the OK button. The dialog box will be replaced by a plot of time histories of the selected variables. The variables will be plotted with default settings. A time history plot of discharge and head is shown in Figure 13.

Features of the plot may be modified using the **Options** menu**.** The **Options** menu item contains seven sub-menu items:

Title creates a dialog box for reviewing and modifying the main title and sub-title

X axis creates a dialog box for reviewing and modifying the x axis title, minimum and maximum x axis values, and the x axis tick interval

Y axis creates a list of all y axes in use in the current plot (one for each variable selected for plotting). Select one to create a dialog box for reviewing and modifying the axis, title, minimum and maximum axis values, and axis tick interval.

Series... menu listing all the variables in use in the current plot

Legend toggle for including the legend box or not

Legend Inside toggle for location of legend inside or outside plot

All creates a dialog box displaying all of these selections.

The legend will be displayed only if the Legend toggle is on (as indicated by a check). The default location for the legend is the right side of the plot. The legend can be moved by clicking and dragging the legend to the desired location. The legend cannot be dragged across plot boundaries, i.e., if the legend is currently inside the plot area, it cannot be dragged outside the plot area and vice versa. Instead, use the Legend Inside menu item to toggle the location of the legend between inside and outside the plot area.

Machine Characteristics

Machine characteristics are plotted from an .INP file. After the file is opened, clicking on the Data Select menu item creates a menu containing two items: Machine and Plot Variables. Upon first opening a file, only Machine will be available for selection. Only after specifying a machine can Plot Variables be selected. Clicking on Machine creates a dialog box that lists machine elements (valves, pumps, pump-turbines, and turbines) available for plotting. Select a machine by clicking on the name of the machine desired. Only one machine element can be selected at a time. Click on the OK button to accept the selection and plot the machine characteristics with default plot settings.

If a different combination of variables is desired, click on the **Plot Variables** item under **Data Select.** The resulting dialog box lists variables available for plotting on the Horizontal and Vertical Axes, as well as a Contour Variable. Variable selection is subject to two constraints: (1) variables cannot be selected for plotting on more than one axis, and (2) the two variables which form the bodies of the characteristics tables cannot be plotted together, e.g., head ratio and torque ratio for pumps and model discharge and power for turbines. Figure 14 shows a plot of Head Ratio as a function of Discharge. Speed Ratio is the contour variable. Also, note the **Options** menu is displayed.

Options available for machine characteristic plots are identical to those available for time history plots with the following exception: since only one variable can be plotted on the y axis, clicking on the **Y Axis** menu item will lead directly to the Y axis dialog box as described above for time history plots.

Error Messages

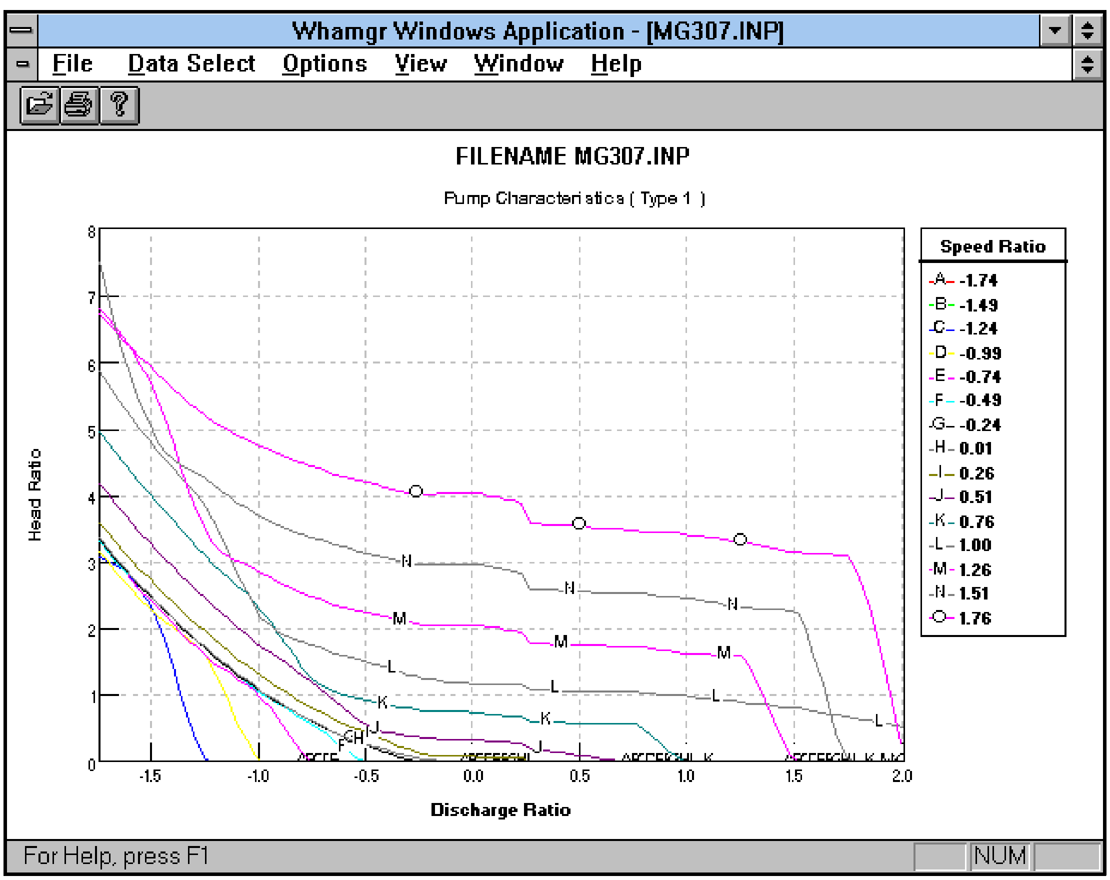


Figure 14. WHAMGR display of pump characteristic curves from .INP file.

During each WHAMGR session, a CHAR.LOG file is created. WHAMGR error messages are directed to this file. If the user experiences difficulty creating a plot, error messages contained in CHAR.LOG should provide some indication of the cause of the difficulties.

3.7.4 Printing

After a plot has been created on the screen, clicking on the **File** menu item will display the following submenu items related to printing:

Print Setup Clicking on this item will create a dialog box for reviewing and modifying printer specifications such as the type of printer, printer part connections, paper size, paper orientation (landscape or portrait), etc.

Print Preview Clicking on this item will cause a preview of the hardcopy output to be displayed on the screen.

Print Clicking on this item causes the current plot to be sent to the printer.

The File menu also includes an Exit item. Clicking on this causes WHAMGR execution to terminate.

3.7.5 View, Window, and Help Menu Items

These menu items are not directly related to making plots with WHAMGR. They are described briefly below.

View View controls the appearance of the Toolbar located at the top of the window and the Status bar located at the bottom of the window. Each of these can be toggled on and off by clicking.

Window The Window menu item controls window arrangements. It is typical of Windows applications and contains the following four menu items: **New, Cascade, Tile,** and **Arrange Icons.** New opens a new window of the current file. Both **Cascade** and **Tile** arrange multiple windows. **Cascade** places the windows in an overlapping arrangement and **Tile** places the windows in a non-overlapping arrangement. **Arrange Icons** arranges icons at the bottom of the screen.

Help Self-explanatory.

4 Command Reference

As stated previously, each primary command introduces a series of related secondary commands which accompany the numeric (and sometimes alphanumeric) data required for simulation. A list of the secondary commands associated with each primary command is given below. The symbol in the "Data" column next to the command indicates the type of data required. The following conventions are used:

a - alphanumeric data required
i - integer value
r - real value
rl, r2, ... multiple real values can follow a single tag

If left blank, no data is required with that secondary command. For some data, the program fixes a default value in the event that the user specifies no value. Default values are shown in the glossary for the appropriate commands.

4.1 Element Commands

The element commands define the various components which make up the hydraulic system. The commands are similar in the use of a four character alphanumeric name for each element. This name, specified by the ID secondary command, must be the first secondary command to appear in the element command string.

Up to 300 elements may be defined for a single system. (Not including characteristics "types," etc.)

The element commands are described on the following pages in alphabetical order, including the commands for machine and valve characteristics (PCHAR, PTCHAR, TCHAR, and VCHAR).

CONDUIT

The properties of an individual conveyance element are input as shown below. There will be a CONDUIT command for each conduit element of the system.

Secondary Commands	Data	Comments
Identifier		
ID	a	User-defined four-character identifier of the conduit. It must be the first secondary command.
Geometric Data		
LENGTH	r	Total length of the conduit in feet.
DIAMETER	r	Diameter of conduit in feet.
THICKNESS	r	Wall thickness of pipe in inches.
ELASTICITY	r	Young's modulus of elasticity for the pipe material in psi.
CELERITY	r	Pressure wave celerity within the conduit in feet per second. This may be specified as an alternative to specifying THICKNESS and ELASTICITY, particularly where the conduit is a tunnel.
FRICTION	r	Darcy-Weisbach friction factor.
Computational Data		
NUMSEG	i	Number of computational segments within the conduit. This defines x in the finite difference equations. In most cases, x should not exceed about 500 feet, though larger values are some times appropriate.
Entrance and Exit Loss Data		

Hydraulic losses at a pipe intake or exit may be specified at the boundary between a CONDUIT element and a RESERVOIR element. The command specifies the loss using a multiplicative coefficient which is applied to the velocity head in the pipe. The applicable subcommands are:

ENDLOSS		Command to indicate that entrance or exit losses are to be computed for this conduit.
AT	a	The identifier of the reservoir element to which the conduit is joined where the loss will take place.
CPLUS	r	The head loss coefficient for flow in the assumed positive direction.
CMINUS	r	The head loss coefficient for flow in the assumed negative direction.

The head loss coefficients default to 0.0 if not specified. Only one ENDLOSS may be specified for a conduit.

Added Loss Data

Added losses may be used to model orifices, bends, and other local geometries in a pipe which introduce head losses in addition to conduit friction. The command specifies the loss via a multiplicative coefficient which is applied to the velocity head. The applicable sub-commands are:

ADDEDLOSS		Command to indicate that added losses are to be computed in the conduit.
AT	r	The location, in feet downstream from the upstream end of the conduit, where the loss is to be included.
CPLUS	r	The head loss coefficient for flow in the assumed positive direction.
CMINUS	r	The head loss coefficient for flow in the negative direction.

The head loss coefficients default to 0.0 if not specified.

Up to 3 ADDEDLOSS commands may be included per conduit. If more are needed, the conduit must be subdivided. Each loss will be specified by a complete set of subcommands above. To specify an added loss at a dummy conduit location (see below), the AT command need not be included but a diameter must be specified in order that a velocity head can be calculated.

Variable Diameter Conduit Data

For a non-prismatic conduit whose diameter varies along its length, the DIAMETER command is replaced by a specified cross-sectional area versus distance relationship. The data are given by alternating DISTANCE and AREA commands. The program will interpolate values of area intermediate to those specified.

VARIABLE		Command to indicate that the conduit has a varying diameter.

The next two commands must be input in pairs.

DISTANCE or D	r	Distance in feet from the upstream end of the conduit. The first value must be 0.0 with subsequent values in increasing order.
AREA or A	r	Cross-sectional area of the conduit (square feet) at the distance specified.

No more than 20 variable diameter conduits may be included in a single system. Computations for variable diameter conduits are based on the average end area of each segment (see Chapter 6), so the user may often wish to specify more segments than for constant diameter conduits.

Dummy Conduits

The dummy conduit is a connector included to give the user flexibility in constructing systems. Generally, the head and discharge at the two nodes which it joins are equated. Four common uses are listed as follows:

1. Temporarily replace another element.

2. Link a boundary element (including surge tank) to a junction. The dummy element must intervene because the connectivity logic will not accept single element branches.

3. Link any element other than a conduit to a Tee Junction.

An added loss may be included as part of a dummy conduit. A diameter must be specified for application 3 above.

DUMMY	Indicates that a conduit is a dummy conduit

Miscellaneous Data

AS a	The AS command is used to request that data from conduit "a" which has already been defined, be copied and used for the current conduit. These data will be altered by subsequent secondary commands.
FINISH	This must be the last command of the conduit input data set.

Examples: CONDUIT Command

```
CONDUIT ID C1 LENGTH 46. DIAMETER 17.0 THICKNESS 0.5
    ELASTICITY 30000000 FRICTION 0.008. FINISH

CONDUIT ID C3 AS C1 LENGTH 500
    NUMSEG 2 FINISH

CONDUIT ID C3 LENGTH 3280. DIAMETER 20.0
    CELERITY 4000. FRICTION .008 ENDLOSS AT RES1
    CPLUS 0.5 CMINUS 1.0 ADDEDLOSS AT 300.
    CPLUS 0.4 CMINUS 0.4 NUMSEG 8 FINISH

CONDUIT ID C4 VARIABLE DISTANCE 0.0
    AREA 314. D 30 A 227. LENGTH 30.
    CELERITY 4000. FRICTION .008 FINISH

CONDUIT ID DUMMY DIAMETER 17.0
    ADDEDLOSS CPLUS 0.4 CMINUS 0.4 FINISH
```

DCHANGE

The DCHANGE command allows the user to specify head loss coefficients for locations in the system where there is a significant change in conduit diameter (Figure 15). The command uses the common convention that the head loss coefficient is applied to the velocity head ($v^2/2g$) of the smaller diameter conduit. For convenience, the more descriptive primary commands EXPANSION or CONTRACTION may be used as alternative to DCHANGE. Since all cases must consider reversed flow, the three commands are entirely synonymous. The secondary commands are:

Secondary Commands	Data	Comments

Identifier

ID	a	User defined identifier, a four character alphanumeric word. This must be the first secondary command.

Head Loss Characteristics

The head loss characteristics of the diameter change may be specified directly, or computed from the geometry by the algorithm presented in Chapter 6. The alternative commands are shown below.

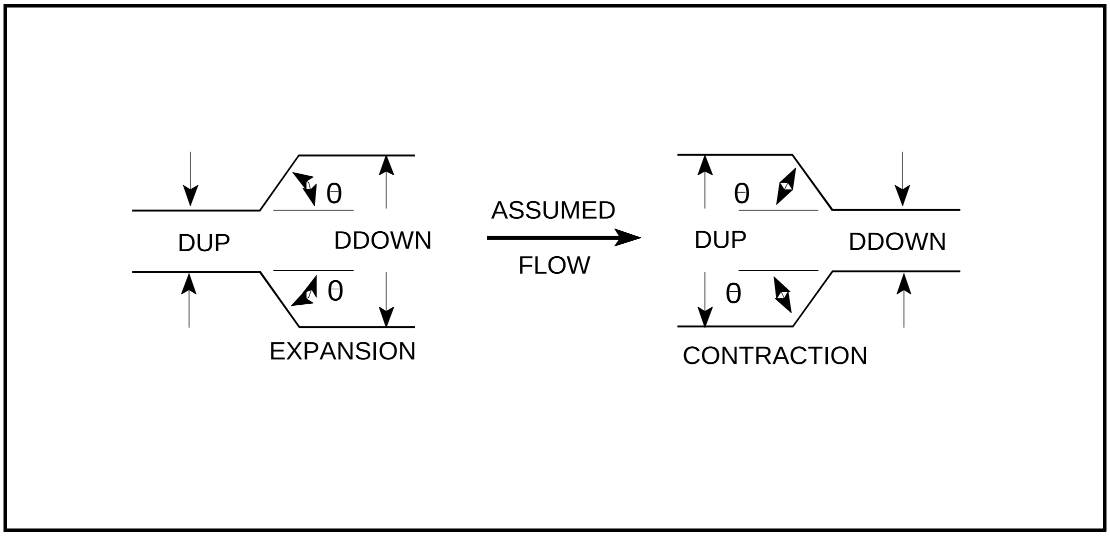


Figure 15. Definition sketch for diameter change elements.

Coefficient Specification

CPLUS	r	Head loss coefficient for flow in the assumed positive direction.
CMINUS	r	Head loss coefficient for flow in the assumed negative direction.

If a CPLUS or CMINUS value is not specified, it will be computed from the geometry. The default is not 0.0.

Geometric Data

DUP	r	The pipe diameter upstream of the diameter change (feet). This is required data only if the upstream element is not a conduit or surge tank.
DDOWN	r	The pipe diameter downstream of the diameter change (feet). This is required data only if the downstream element is not a conduit or a surge tank.
ANGLE	r	The angle ⊖ shown in the definition sketch (degrees). This is required only if the head loss coefficients are to be calculated. The default value is 90°.

Miscellaneous Command

FINISH		The last command of the input stream.

Examples: DCHANGE Command

DCHANGE ID DC1 FINISH

The coefficients for DC1 will be computed from the geometry using the default value of the flare angle.

DCHANGE ID DC2 CPLUS 0.4 CMINUS 0.2 FINISH

FLOWBC

This command is used to specify a flow boundary condition. Such a condition may be used as a substitute for a detailed model of a portion of the hydraulic system.

Secondary Commands	Data	Comments
Identifiers		
ID	a	User defined identifier. This must be the first secondary command.
Boundary Condition Data		
Q	r	Constant discharge at the boundary in cfs.
QSCHEDULE	i	Specifies the number of a time variant discharge schedule defined with a SCHEDULE command.
Miscellaneous Command		
FINISH		The last secondary command of the input string.

Examples: FLOWBC Command

```
FLOWBC   ID   FBC1   Q   3820.   FINISH
FLOWBC   ID   FBC2   QSCHEDULE   1   FINISH
```

GOVERNOR

WHAMO includes algorithms for mechanical and electrical governing mechanisms for turbines. These algorithms model wicket gate movements and speed variations during minor load variations, as specified with the GOVERN option under the OPTURB or OPPT command. For load rejection or major load adjustments, the governor equations may not be applicable. In these cases, wicket gate operation should be specified with the SCHEDULE command.

Governors are modeled using equation (6.37) of Section 6.5.

$$a_1 \frac{d^3G}{dt^3} + a_2 \frac{d^2G}{dt^2} + a_3 \frac{dG}{dt} + a_4 \, (G - G_r) = a_5 \frac{d^2\omega'}{dt^2} + a_6 \frac{d\omega'}{dt} + a_7 \, (\omega') + a_8$$

For electrical governors, ῶ' includes a generation feedback term as well as speed variation.

Secondary Commands	Data	Comments

Identifier

ID	a	Identifier of the turbine or pump-turbine element which is to be controlled by this governor unit.

Mechanical Governors

The coefficients of equation (6.37) may be input directly for mechanical-type governors without power generation response. The following subcommands are used. In each case, the default value is zero.

Governor Characteristics

AONE	r	The coefficient a_1 in Equation (6.37).
ATWO	r	The coefficient a_2 in Equation (6.37).
ATHREE	r	The coefficient a_3 in Equation (6.37).

AFOUR	r	The coefficient a_4 in Equation (6.37).
AFIVE	r	The coefficient a_5 in Equation (6.37).
ASIX	r	The coefficient a_6 in Equation (6.37).
ASEVEN	r	The coefficient a_7 in Equation (6.37).
AEIGHT	r	The coefficient a_8 in Equation (6.37).

Alternatively, a set of parameter values commonly associated with mechanical governors may be input. Coefficients of Equation 6.37 are automatically computed from these parameters according to relations given in Section 6.5. Typical parameter values are given in Section 6.5.

TG	r	Governor promptitude time constant (seconds).
TR	r	Dashpot relaxation time constant (seconds).
PDROOP	r	Permanent speed droop (per unit).
TDROOP	r	Temporary speed droop (per unit).

Electric Governors

The coefficients of Equation 6.37 may also be input directly for electric-type governors with power generation response. The same subcommands, AONE through AEIGHT, are used, but the ELEC and THETA subcommands must also be entered. Also, RPOWER should be input if the initial power is not to be taken as the reference power.

ELEC		Input in conjunction with the governor equation coefficients (AONE through AEIGHT) to indicate an electric-type governor.
THETA	r	Speed regulation term. This is the multiplicative factor applied to per unit power generation in the $\tilde{\omega}'$ term of Equation 6.37 for electric governors. It is similar to speed droop except response is to generator power output. Typical value is 0.05.

RPOWER	r	Reference power output (HP) used to normalize power into per unit terms in the electric governor equation. Typically this is the turbine rated power. Defaults to the specified initial power output in isolated load simulations.

Electric governor characteristics may also be specified using the following parameters. These are defined in Section 6.5 and typical values are given there. The ELEC command need not be included when these parameters are used.

KP	r	Proportional gain.
KI	r	Integral gain (1/seconds).
KD	r	Derivative gain (seconds).
THETA	r	Speed regulation term.
TT	r	Pilot servomotor time constant (seconds). Typical values range from 0.05 to 0.1.
RPOWER	r	Reference power (HP). Defaults to the specified initial power output in isolated load simulations.

Default Governor Characteristics

Default governor characteristics will be computed using the relations given in Section 6.5. It is necessary only to specify the water starting time and to indicate whether a mechanical or electrical governor is to be modeled. The reference power may also be specified. This value will be used to compute the mechanical starting time, and also to normalize the power term in the electric governor equation.

TW	r	Water starting time (seconds), as defined in Equation 6.42, used to compute default governor characteristics. Compute TW from the turbine to the nearest surgetank or reservoir. Do <u>not</u> include this command if default values are not to be used.
ELEC		Indicates that an electric-type governor is to be modeled.

MECH		Indicates that a mechanical-type governor is to be modeled. (Default)
RPOWER	r	Reference power (HP). Defaults to the specified initial power in isolated load simulations.

Governor Limits

If desired, maximum wicket gate opening and closing rates may be specified. Though discontinuities of governor behavior may not be completely accounted for, gate movement during simulation will not be allowed to exceed the specified rates.

OPENMAX	r	Maximum rate of wicket gate movement to a higher percent valued position in percent per second.
CLOSEMAX	r	Maximum rate of wicket gate movement to a lower percent valued position in percent per second.

Miscellaneous Command

AS	a	(See CONDUIT)

Terminating Command

FINISH

Examples: GOVERNOR Command

```
GOVERNOR    ID TUR1  MECH     TW     1.0     FINISH

GOVERNOR    ID PT1   TG       0.15   TR      5.8
     PDROOP 0.05     TDROOP   0.1    FINISH

GOVERNOR    ID TURB  KP       1.0    KI 0.5  KD    0.8
     TT 0.05   THETA   0.05   FINISH
```

ONEWAY

This element acts as a simple non-return valve, but it does not simulate a real non-return valve where reverse flow may sometimes occur, causing valve slamming and pressure surges. No reverse flow is allowed with a ONEWAY element. Forward flow is unrestricted, but a head loss coefficient may be applied.

In order to prevent significant reverse flow during simulation, the program may sometimes compute at a time step shorter than specified. If this causes computational difficulties, the user can specify a minimum time step with the DTMIN command under CONTROL.

Normally, flow in the positive direction is allowed and flow in the negative direction is restricted. This convention can be reversed by entering a REVERSE command.

Secondary Commands	Data	Comments
Identifier		
ID	a	User-defined; must be first secondary command.
Physical Data		
DIAMETER	r	Valve diameter in feet.
CLOSS	r	Head loss coefficient for flow in the allowed direction.
Miscellaneous Command		
REVERSE		Specifies that flow is allowed in the assumed negative direction and restricted in the positive direction. If this command is not entered, flow is restricted in the negative direction.
Terminating Command		
FINISH		

Example: ONEWAY Command

ONEWAY ID CHEC DIAM 2.5 CLOSS 0.8 FINISH

PCVALVE

This element simulates pressure control valves. Like VALVE elements, the flow characteristics are defined to be of a particular type, either pre-programmed or input elsewhere with a VCHAR command. Unlike VALVE elements for which a schedule of valve opening is input, valve opening and closing rates are specified as a function of pressure at one or two reference points in the system.

The PCVALVE element can be used to represent pressure relief, pressure control, differential pressure control, and flow control valves as shown in the examples.

Note that this is not a boundary element. There must be a boundary element or junction at the ends of branches containing a PCVALVE element.

Secondary Commands	Data	Comments
Identifier		
ID	a	User-defined identifier. This must be the first secondary command.
Geometric Data		
DIAMETER	r	Diameter of the valve in feet.
Flow Characteristics		
GATE		Specifies that the valve is a disk type gate valve.
BUTTERFLY		Specifies that the valve type is a butterfly valve.
HOWELL		Specifies that the valve type is a Howell-Bunger valve.
SPHERICAL		Specifies that the valve type is a spherical valve.
TYPE	i	The flow characteristics for the valve are defined by TYPE i under a VCHAR command. The characteristics are discharge or head loss versus valve position.

Reference Nodes

The valve responds to computed pressure at one or two specified reference nodes in the system. If only one reference node is specified then the valve will respond to the computed pressure at that node as described below under Pressure Control. If two reference nodes are specified, then the valve responds to the difference between the pressures at the two nodes. This difference is defined as the pressure at the first reference node specified minus the pressure at the second reference node.

REFERENCE		Indicates that specification of reference node(s) follows
NODE	i	Reference node number

The elevation of reference nodes must be entered with the SYSTEM primary command so that pressure may be computed at those locations.

Pressure Control

A target pressure is defined at which the valve will tend to neither open nor close, no matter what the current valve position is. This pressure refers to the gage pressure in psi at the reference node if only one reference node is specified. If there are two reference nodes, then this refers to the difference in gage pressures at the nodes as described above.

Note that if a PCVALVE is at a limit, either fully open or fully closed, it will not necessarily move even if the computed pressure differs from the target pressure. For example, a closed PCVALVE configured as a pressure relief valve will remain closed as long as the computed pressure is less than the target.

PTARG	r	Target pressure (psig)

For both the opening and closing direction, a maximum rate of movement, and the pressure (or pressure difference) at which that rate is achieved, must be specified. If the computed pressure (or pressure difference) is between the target pressure (corresponding to zero valve movement) and the pressure at maximum rate, then the valve movement rate will be interpolated as either a linear or square root function as specified by the user.

OPENING		Indicates that the PMAX, MAXRATE, and interpolation specification commands which follow apply to valve <u>opening</u>.
CLOSING		Indicates that the PMAX, MAXRATE, and interpolation specification commands which follow apply to valve <u>closing</u>.
MAXRATE	r	Maximum rate of valve movement expressed as the time of valve travel in seconds from closed to fully open or fully open to closed.
PMAX	r	Pressure or pressure difference (psig) at which the maximum rate of valve movement is achieved.
LINEAR		Indicates that a linear interpolation function will be used to determine the valve movement rate when the pressure is between PTARG and PMAX (Default).
ROOT		Indicates that a square root interpolation function will be used to determine the valve movement rate when the pressure is between PTARG and PMAX.

Note that if PMAX for valve opening is greater than PTARG then PMAX for valve closing must be less than PTARG. Similarly, if PMAX for opening is less than PTARG, PMAX for closing must be greater.

Terminating Command

FINISH

Examples: PCVALVE Command

Pressure Relief Valve - For a pressure relief valve, a single reference node upstream of the valve is specified. PTARG will be the pressure at which the valve opens. PMAX for opening will be greater than PTARG and PMAX for closing will be less than PTARG.

```
PCVALVE ID RELF GATE DIAM 0.8
    REFERENCE NODE 100    PTARG 200.
    OPENING MAXRATE 30.   PMAX 210.   LINEAR
    CLOSING MAXRATE 30.   PMAX 190.   LINEAR
    FINISH
```

Pressure Reducing Valve - For a pressure reducing valve a single reference node downstream of the valve is specified. PTARG will be the desired downstream pressure. PMAX for opening will be less than PTARG and PMAX for closing will be greater than PTARG.

```
PCVALVE ID REDU BUTTERFLY DIAM 1.0
    REFERENCE NODE 200    PTARG 100.
    OPENING MAXRATE 60.   PMAX 75.    ROOT
    CLOSING MAXRATE 60.   PMAX 125.   ROOT
    FINISH
```

Differential Pressure Control Valve - In this case, two reference nodes are specified, one on each side of the valve. The reference node on the upstream side of the valve is specified first for convenience, i.e., to work with positive numbers. PTARG is the desired pressure difference between the two points. PMAX for opening will be greater than PTARG and PMAX for closing will be less than PTARG.

```
PCVALVE ID DIFF BUTTERFLY DIAM 1.2
    REFERENCE NODE 100    NODE 200    PTARG 50.
    OPENING MAXRATE 60.   PMAX 65.    ROOT
    CLOSING MAXRATE 60.   PMAX 40.    ROOT
    FINISH
```

Flow Control Valve - For a flow control valve, two reference points are specified, both either upstream or downstream of the valve. The upstream reference node is specified first for convenience. PTARG will be the pressure difference between the two points which corresponds to the desired flow rate. PMAX for opening will be less than PTARG and PMAX for closing will be greater than PTARG.

```
PCVALVE ID FLOW BUTTERFLY DIAM 0.75
    REFERENCE NODE 50     NODE 100    PTARG 10.
    OPENING MAXRATE 25.   PMAX 6.  ROOT
    CLOSING MAXRATE 25.   PMAX 14.    ROOT
    FINISH
```

P-T

The identifier and specifications of each pump-turbine are input using the P-T command. Pump-turbine characteristic tables will be input separately using the PTCHAR command, while the operation of this pump-turbine will be specified using the OPPT command.

Secondary Commands	Data	Comments
Identifier		
ID	a	User defined identifier. This must be the first secondary command.
TYPE	i	i is a user defined number designating which set of P-T characteristics defined under PTCHAR corresponds to this pump-turbine. No more than 6 pump-turbine types may be specified for a system. (Defaults to 1)
Pump-Turbine Specifications		
FRICTION	r	Friction loss at synchronous speed in H.P. (Default to 0.0)
WINDAGE	r	Windage loss at synchronous speed in H.P. (Default to 0.0)
WR2	r	Rotational inertia, WR^2, in lb-ft.^{2}.
SYNCSPD	r	Synchronous speed in RPM when the same in both directions.
PSYNC	r	Synchronous speed in the pumping direction in RPM.
TSYNC	r	Synchronous speed in the turbine direction in RPM.
DIAMETER	r	Diameter of the runner/impeller in feet.

Miscellaneous

AS	a	(See CONDUIT)
FINISH		

Example: P-T Command

```
P-T  ID        PTI      TYPE 1
     WR2    66000000.     SYNCSPD     300.
     DIAMETER   10.545      FINISH
```

Default values (=0.0) of friction and windage are used and the synchronous speed is the same in both directions.

PTCHAR

The characteristics of each pump-turbine TYPE are input with a PTCHAR command in the form of tables derived from model test data. These test data apply to a model of one foot diameter under one foot of driving head. The input includes lists of operating points defined by speed, discharge, and torque which apply at specified gate positions — one list for each gate position.

To simplify computations and still accommodate all portions of typical pump-turbine curves, the program stores the data in terms of a radial coordinate system of speed vs. discharge. It is necessary, therefore, that the data points input be selected at the intersection points of the gate curves and a set of radial lines drawn on a speed-discharge plot, such as shown in Figure 16.

Suggested input steps are:

(1) Draw a set of radial lines on a speed-discharge plot such that the intersection points adequately define all of the gate curves over the expected range of machine operation.

(2) Begin with the highest gate position to be tabulated (usually 100 percent). Enter the value with a GATE subcommand.

(3) Begin with the most extreme radial in the normal pump quadrant (i.e., the radial most removed from the negative speed axis and closest to the negative discharge axis). Record the speed and discharge values at the intersection with the gate curve.

(4) Use the torque-speed characteristics plot to determine the torque at the operating point defined by (3).

(5) Enter the speed, discharge, and torque values for this radial and gate with the NQT sub-command.

(6) Repeat steps (3) through (5) for each radial in sequence moving counterclockwise.

(7) Repeat steps (2) through (6) for each gate value in sequence down to 0 gate (in most cases).

Note in Figure 16 that the first radial (indicated by the dashed line) intercepts only the curve for gate G5. For gates G1-G4, the user should enter NODATA instead of the NQT subcommand to indicate that data are unavailable for this radial.

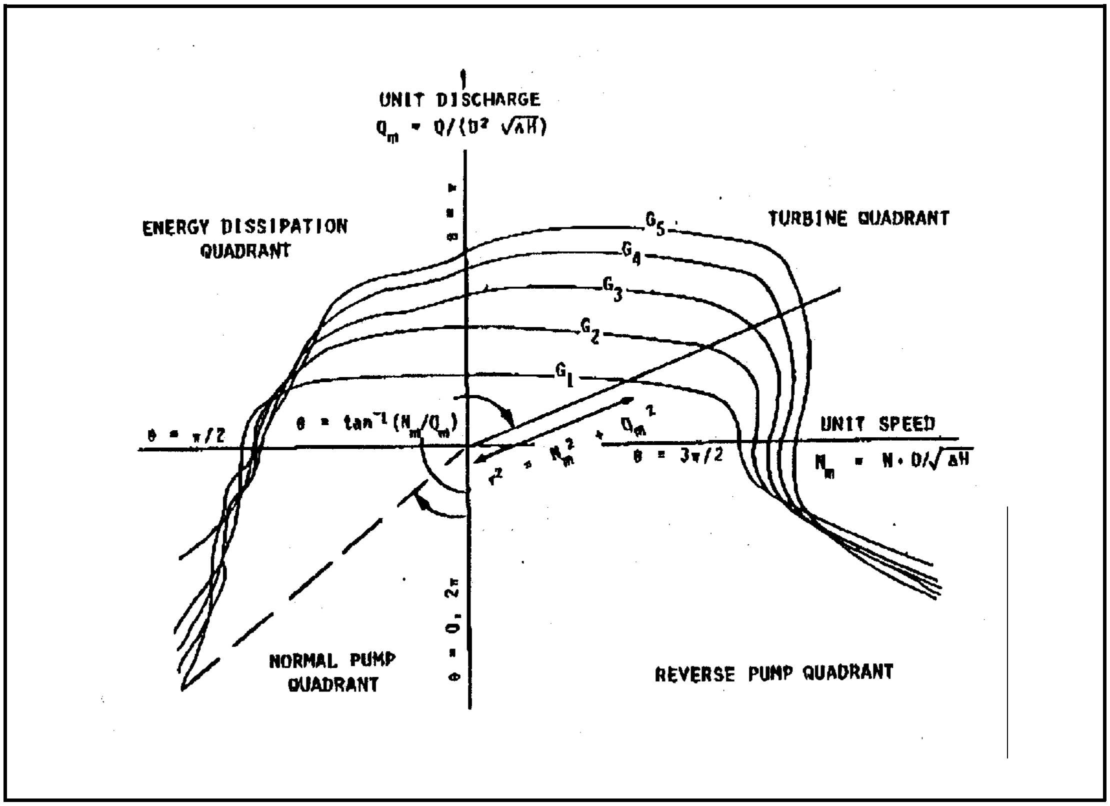


Figure 16. Speed-Discharge characteristics plot for Pump-Turbines.

More than one pump-turbine in a system may be of the same TYPE (i.e., for which the same characteristics apply). Up to 6 sets of pump-turbine characteristics may be input for a single system.

Secondary Commands	**Data**	**Comments**
Identifier		
TYPE	i	i corresponds to a TYPE number specified under P-T. This must be the first secondary command.

Characteristics

Up to 50 gate positions (including 0 gate) may be specified in ascending or descending order.

GATE	r	Specified wicket gate position (percent opening). The data which follow apply for this gate position.

For each gate position, up to 50 operating points may be specified. Each operating point given corresponds to the intersection of the gate curve and a radial line drawn on a speed-discharge plot. Referring to Figure 16, the data are given for radials in clockwise sequence beginning in the normal pump quadrant. Data must be entered for the same radials in the same order. The total number of gate positions multiplied by the number of radials cannot exceed 625.

NQT	r_1, r_2, r_3...	Operating point on characteristics curve. r1 is a model speed (rpm). r2 is a model discharge (cfs). r3 is model torque (ft-lbs). Positive speed and discharge are in the turbine (generating) direction. Positive torque is also in the turbine direction (resists pumping, impels generating).
NODATA	i	Enter this command instead of NQT to indicate that data is not available for the current radial position and GATE. i is the number of consecutive radials for this gate for which data is unavailable (Defaults to 1).
ZERO	i	Enter this command instead of a set of NQT commands under GATE 0.0 if all speed, discharge, and torque values are zero. i is the number of radial positions in the table.

Terminating Command

FINISH

Example: PTCHAR Command

Referring to Figure 16, assume that data is to be entered for 6 gate positions (including 0 gate) and only 4 radials as shown (this is unrealistic). The form of the input commands would be as follows.

```
PTCHAR TYPE 1
GATE 100
```

```
    NQT   -100.0 -2.0    5.3
    NQT   - 25.2  1.0    3.4
    NQT   95.7    0.8    4.8
    NQT   121.2  -0.9   -3.6
GATE 80
    NODATA 1
    NQT   - 23.6  0.9    3.0
    NQT   92.0    0.75   3.1
    NODATA 1
GATE 60
    NODATA 1
    NQT   - 20.7  0.75   2.6
    NQT   83. 2   0.68   2.4
    NODATA 1
GATE 0
    ZERO 4
FINISH
```

Pump-Turbine Characteristics - Old Method

The first version of WHAMO used different conventions for pump-turbine characteristics input. Tables were input of model discharge versus speed and gate position and model torque versus speed and gate position. This method proved unsatisfactory because in many cases multiple values of discharge or torque can exist at a given speed and gate. For this reason, the radial method described above was adopted.

The present version of WHAMO will accept input and make computations based on the old representation of characteristics. The following commands are entered in the old method:

Secondary Commands	Data	Comments
TYPE		Same as above
DMODEL		Diameter (inches) of model for which the characteristics apply. (Default to 12 inches).
GATE		Values of wicket gate positions (in percent open) which head the columns of the characteristics table.

SPEED	Values of model speed (RPM) to head the rows of the characteristics tables.
Q	A table of model discharge values (cfs) corresponding to the specified GATE and SPEED values. These must be listed in row order. (GATE values varied over constant SPEED values.)
TORQUE	A table of model torque values (ft-lbs) corresponding to the specified GATE and SPEED values.
FINISH	

PUMP

The identifier and specifications of each pump are input using the PUMP command. Pump characteristics tables will be input separately using the PCHAR command, while the operation of the pumps will be specified using the OPPUMP command.

Secondary Commands	Data	Comments
Identifiers		
ID	a	User defined; must be first secondary command.
TYPE	i	i is a user defined number designating which set of characteristics defined under PCHAR corresponds to this pump. No more than 6 pump TYPEs may be specified for a system.
Pump Specifications		
RHEAD	r	Rated head in feet.
RQ	r	Rated discharge in cfs.
RSPEED	r	Rated speed in rpm.
RTORQUE	r	Rated torque in lb-ft.
WR2	r	Rotational inertia, WR^2 , in lb-ft^2.
Miscellaneous Commands		
AS	a	(see CONDUIT)
FINISH		

Example: PUMP Command

```
PUMP    ID    P1     TYPE     1     RHEAD    150.    RQ       300.
RSPEED    450.    RTORQUE    47670    WR2      250355.
FINISH
```

PCHAR

The dimensionless characteristics of each pump TYPE are input with a PCHAR command in the form of two tables which express ratios of actual parameter values to specified rated values. The input includes four lists: first, lists of the speed ratios and discharge ratios which head the columns and rows of the tables, then lists of head ratios and torque ratios which form the body of the two tables. These lists must be in the particular order described below. Up to six sets of pump characteristics may be input for a single system.

Secondary Commands	Data	Comments
Identifier		
TYPE	i	corresponds to a TYPE number specified under PUMP. There may be several pumps for which the same characteristics apply and therefore have the same TYPE number. This must be the first secondary command.
Pump Characteristics		
SRATIO	rl, r2, r3...	A list of specified speed ratios which head the columns of the pump characteristics tables, where the speed ratio is the actual pump speed divided by the rated pump speed. A "four quadrant" table will include negative speed ratios in the turbine direction as well as positive speeds in the pumping direction. There must be at least 3 and no more than 50 values listed in ascending or descending order.
QRATIO	rl, r2, r3...	A list of specified discharge ratios which head the rows of the pump characteristics tables where the discharge ratio is the actual discharge divided by the rated discharge. A "four quadrant" table will include negative discharges in the turbine direction as well as positive discharges in the pumping direction. There must

be at least 3 and no more than 50 values listed in ascending or descending order.

The SRATIO and QRATIO commands must precede the HRATIO and TRATIO commands within the PCHAR primary commands.

HRATIO	rl, r2, r3...	A characteristic table of head ratios correspond ing to the specified SRATIO and QRATIO values. Head refers to the head difference across the pump. This array must be listed in row order. (SRATIO values are varied over constant QRATIO values.) There may be up to 625 values.
TRATIO	rl, r2, r3...	A characteristic table of torque ratios correspond ing to the specified SRATIO and QRATIO values. Positive torque resists pumping. This array must be listed in the same order as HRATIO. There may be up to 625 values.

Miscellaneous Command

FINISH

Example: PCHAR Command

To illustrate the order in which the data must be input, suppose both the head ratio and torque ratio tables were as shown:

QRATIO	SRATIO				
	1	2	3	4	5
1	1	2	3	4	5
2	6	7	8	9	10
3	11	12	13	14	15

Then the PCHAR command would be as follows:

PCHAR TYPE 1

SRATIO 1. 2. 3. 4. 5.
QRATIO 1. 2. 3.
HRATIO 1. 2. 3. 4. 5. 6. 7. 8. 9. 10.
11. 12. 13. 14. 15.
TRATIO 1. 2. 3. 4. 5. 6. 7. 8. 9. 10.
11. 12. 13. 14. 15.
FINISH

RELIEF

This element simulates pressure actuated valves. It is similar to the PCVALVE element, in that the valve opening is a function of pressure at a reference point in the system. The operating function is defined differently from PCVALVE; a table of valve opening versus pressure is specified.

The valve typically responds to upstream pressure relative to the specified datum (generally the elevation of the valve). As an option, the user may specify that the valve responds to pressure at any particular node point in the system.

Note that this is not a boundary element. There must be a boundary element or junction at the downstream end of branches containing a RELIEF element. Up to 5 different RELIEF elements can be simulated. If more relief elements are required, use the AS command when identical pressure-opening tables apply.

Secondary Commands	Data	Comments
Identifier		
ID	a	User-defined; must be first secondary command.
Geometric Data		
DIAMETER	r	Diameter of the valve in feet.
ELEV	r	Elevation of the valve or datum from which upstream pressure is calculated (feet).
Flow Characteristics		
GATE		Specifies that the valve is a disk type gate valve.
BUTTERFLY		Specifies that the valve type is a butterfly valve.
HOWELL		Specifies a Howell-Bunger valve.
SPHERICAL		Specifies that the valve type is a spherical valve.
TYPE	i	The flow characteristics for the valve are defined by TYPE i under a VCHAR command. The

characteristics are discharge or head loss versus valve position.

Pressure Actuation

A table of upstream pressure versus valve opening must be supplied for each RELIEF element. An ACTUATION command is used, followed by pairs of PRES and OPENING commands.

ACTUATION		Indicates that a table of valve opening versus pressure follows.

At least 2 and up to 15 PRES-OPEN pairs are entered in ascending order of pressure. At pressures less than the first value or greater than the last value, the first or last value of valve opening will apply.

PRES P	r	Gage pressure upstream of the valve in psi relative to the specified elevation. This should precede OPENING.
OPENING or O	r	Gate or valve position in percent opening or degrees depending on the type of valve specified.

Maximum Opening and Closing Rates

The maximum rate of the opening and the maximum rate of valve closing may be specified with ORATE and CRATE commands. If the valve position determined by pressure implies a rate of movement greater than the specified maximum rate, the valve position will be recalculated based on the maximum rate.

ORATE	r	Time of valve travel in seconds from closed to fully open position. Defaults to 0.
CRATE	r	Time of valve travel in seconds from fully open to closed position. Defaults to 0.

Location of Pressure Response

NODE	i	The valve will respond to pressure computed at node i. If this subcommand is omitted, response

		will be to pressure immediately upstream of the valve.
AREA	r	Flow area for computing velocity to determine pressure at reference node. If this subcommand is omitted, the flow area will be computed from the valve diameter.

Error Tolerance

The valve opening at a given time step is estimated based on the pressure computed at the previous time step. If the new computed pressure corresponds to a valve opening significantly different from the estimated opening, the time step length is reduced and the computations repeated. The user may specify the error tolerance. A small tolerance will reduce valve oscillation in some simulations. A large tolerance will reduce CPU time for some simulations. Also, the user may specify the minimum computational time step with the DTMIN subcommand under CONTROL.

TOLERANCE	r	Error tolerance in RELIEF element estimated valve position (percent opening or degrees). Defaults to 2.0.

Miscellaneous Command

AS	a	Same as for CONDUIT command.

Terminating Command

FINISH

Example: RELIEF Command

```
RELIEF   ID   PRV   HOWELL   DIAM   5.0   ELEV   327
ACTUATION
PRES   0.0    OPEN    0.0
P     300    O      0.0
P     325.   O      50.0
P     350.   O      100.0
ORATE  10.    CRATE   30.0
FINISH
```

RESERVOIR

A boundary water surface elevation is defined by the RESERVOIR element.

Secondary Commands	Data	Comments
Identifiers		
ID	a	User defined identifier. This must be the first secondary command.
Boundary Condition Data		
ELEV	r	Constant elevation of the water surface in feet.
HSCHEDULE	i	Specifies the number of a time variant elevation schedule, defined with a SCHEDULE command.
Miscellaneous Command		
FINISH		The last secondary command of the input string.

Examples: RESERVOIR Command

```
RESERVOIR    ID   HW   1124.   FINISH
RESERVOIR    ID   TW   HSCHEDULE   1   FINISH
```

SURGETANK

Either a simple surge tank, one way surgetank, Johnson differential surge tank, or pressurized air chamber type surge tank may be modeled. A simple surge tank may include an outflow port which allows water to leave the system. An air chamber may be vented or unvented.

To simulate a oneway surge tank, it is necessary for the user to also specify a check valve (ONEWAY) element upstream of the surgetank to prevent flow into the surgetank. This is not done automatically by the program. WHAMO conventions for positive flow direction dictate that a REVERSE subcommand must be included with the ONEWAY command specifying the check valve to prevent flow into the surge tank.

Surge tank calculations are done in two parts. First, conduit type computations are made, where the "conduit" length depends on the water surface elevation. Next, the boundary equation at the water surface is developed. Therefore, celerity, or wall thickness and elasticity, and a friction factor must be input as for conduits.

Up to 10 SURGETANK elements may be specified in a simulation.

Secondary Commands	Data	Comments
Identifiers		
ID	a	User-defined identifier. This must be the first secondary command.

The type of surge tank is entered immediately after the identifier. If omitted, a simple surge tank is assumed.

SIMPLE		Indicates that the surge tank is a simple surge tank (Default condition).
ONEWAY		Indicates that the surge tank is a oneway surge tank. Oneway surge tanks in WHAMO function exactly as simple surge tanks, and input specifications are the same, except:

- # The starting water level may be lower than the initial operating head in the pipeline.

- # In this case, the starting water level is specified by the user with the ELTOP subcommand.

DIFFERENTIAL		Indicates that the surge tank is a Johnson differential surge tank.
AIR		Indicates that the surge tank is an air chamber, either vented or unvented.
TWOSTAGE		The 2-stage surge tank capability of the original WHAMO has been retained in this version. It is not further documented here because a single tank of that geometry can now be more accurately modeled with a simple surge tank of variable area.

Physical Data for Simple Surge Tanks and Oneway Surge Tanks

ELBOTTOM	r	Elevation of the bottom of the tank in feet.
ELTOP	r	Elevation of the top of the tank in feet for simple surge tanks, starting water level for oneway surge tanks.
THICKNESS	r	Wall thickness of surge tank in inches.
ELASTICITY	r	Young's modulus of elasticity for the surge tank material in psi.
CELERITY	r	Pressure wave celerity within the surge tank in feet per second. This may be specified as an alternative to specifying THICKNESS and ELASTICITY.
FRICTION	r	Overall Darcy-Weisbach friction factor for the surge tank.

DIAMETER	r	Diameter of the surge tank in feet. Specified when the tank is cylindrical with uniform diameter.
SHAPE		When the tank is not a uniform cylinder this command precedes a table of elevation versus area points which define its geometry.

Pairs of elevation and area values are input in ascending order of elevation, with a maximum of 25 points. The first elevation entered must equal the ELBOTTOM value, and the last elevation entered must equal ELTOP. Areas are assumed to vary linearly between specified elevation points.

ELEV E	r	Elevation in feet.
AREA A	r	Area in square feet.

Outflow Data for Simple Surge Tanks

Surge tank overflow will be computed when the water surface elevation exceeds the top of the tank. The standard weir flow equation, using the circumference at the top of the tank, is applied.

CWEIR	r	Weir coefficient for surge tank overflow calculation (Default is 3.3).

An orifice at the side of the surge tank may be specified which allows outflow from the system depending on the water level. Orifice equations are used (see Section 6.6) which apply when the water surface either partially or completely submerges the outlet. Note that if the water level exceeds the top of the tank, overflow is computed <u>in addition to</u> flow through a side outlet.

OUTLET		Indicates that the commands which follow define an outlet which allows water to leave the system.
ELOUT	r	Bottom elevation of outlet in feet.
DOUT	r	Diameter of outlet if circular (feet).

WOUT	r	Width of outlet if rectangular (feet).
HOUT	r	Height of outlet if rectangular (feet).
CDOUT	r	Orifice discharge coefficient. (Defaults to 0.61. This is equivalent to a weir coefficient of 3.3 for a partially submerged rectangular outlet.)

Physical Data for Differential Surge Tanks

ELBOTTOM	r	Same as for simple surge tanks.
RTOP	r	Elevation of the top of the riser in feet.
TTOP	r	Elevation of the top of the outer tank in feet.
RDIAM	r	Diameter of the riser in feet.
TDIAM	r	Diameter of the outer tank in feet.
THICKNESS	r	Wall thickness of the riser in inches.
ELASTICITY	r	Young's modulus of elasticity for the riser material in psi.
CELERITY	r	Pressure wave celerity within the riser in feet per second. This may be specified as an alternative to specifying THICKNESS and ELASTICITY.
FRICTION	r	Darcy-Weisbach friction factor for the riser.
CWEIR	r	Weir coefficient for riser overflow calculation (Default is 3.3).
AORIFICE	r	Area in square feet of the orifice between the riser and the outer tank.
CDIN	r	Orifice coefficient of discharge for flow entering the outer tank from the riser. Represented by C_d in the equation:

$$Q = C_D A \sqrt{2g\ H}$$

CDOUT	r	Orifice coefficient of discharge for flow leaving the outer tank to the riser.

Physical Data for Air Chamber Surge Tanks

ELBOTTOM	r	Elevation of the bottom of the tank in feet.
ELTOP	r	Elevation of the top of the tank in feet.
THICKNESS	r	Wall thickness of surge tank in inches.
ELASTICITY	r	Young's modulus of elasticity for the surge tank material in psi.
CELERITY	r	Pressure wave celerity within the surge tank in feet per second. This may be specified as an alternative to specifying THICKNESS and ELASTICITY.
FRICTION	r	Darcy-Weisbach friction factor.
DIAMETER	r	Diameter of the tank in feet. Specified when the tank is an upright cylinder with uniform diameter.
SHAPE		When the tank is not a uniform cylinder, this command precedes a table of elevation versus area points which define its geometry.

Pairs of elevation and area values are input as described for a simple surge tank.

ELEV E	r	Elevation in feet.
AREA A	r	Area in square feet.

Thermodynamic Data for Air Chamber Surge Tanks

TEMP	r	Ambient temperature in °F. This is assumed to be the initial internal temperature as well as the constant external temperature (Defaults to 68 °F).
PBAR	r	Ambient barometric pressure in psi (Defaults to 14.7 psi).

Two alternative methods, described in Section 6.6, are used for computing the air pressure-volume relation in the tank. These are the polytropic gas equation and rational heat transfer (RHT) methods.

N	r	Exponent in the polytropic gas equation $PV^n = K$. The allowable range is 1.0 to 1.4 (Defaults to 1.25).
RHT		Indicates that the RHT method is used. If this is not entered, the polytropic gas equation is adopted.

Initialization for Air Chamber Surge Tanks

In order that the initial air chamber conditions can be computed, the user must supply either the initial air pressure, initial water surface elevation, or the initial air mass. One, and only one, of the three options must be selected.

PINIT	r	Initial absolute air pressure in the chamber in psi.
WSINIT	r	Initial water surface elevation in the air chamber in feet.
AINIT	r	Initial air mass in the chamber in terms of cubic feet at standard temperature and pressure (68 °F, 14.7 psi).

Venting for Air Chamber Surge Tanks

Air chambers may be vented by open orifices, pressure relief valves, or other means. Air flow is modeled using equations (6.85) and (6.86) in Section 6.6. The orifice discharge coefficient, C, multiplied by the area, A, is entered as a function of pressure in the tank. If no table is entered, it is assumed that no air is allowed into or out of the chamber.

VENT		Indicates that a table follows of discharge coefficient times area versus pressure.

Following the VENT sub-command, up to 15 PRES-CA pairs are entered in ascending order of pressure. At pressure less than the first value or greater than the last value, the first or last value of CA will apply. A single pair may be entered.

PRES	r	Absolute air pressure in the chamber in psi. This should precede CA.
CA	r	Orifice discharge coefficient multiplied by the orifice area (inches). This is C*A in equations (6.85) and (6.86) in Section 6.6.

Terminating Command

FINISH

Examples: SURGETANK Command

```
SURGETANK   ID  ST   ELBOTTOM  1515.
ELTOP   1660.  DIAM  70.    CELERITY  2840.
FRICTION  0.02   FINISH
```

A simple surge tank is specified above by default.

```
SURGETANK   ID  OWST    ONEWAY
      ELBOTTOM  232.  ELTOP  238.  DIAM  25.
      CELERITY 3000.   FRICTION 0.012  FINISH
```

To simulate oneway surge tank as indicated above, the user must also specify a check valve element upstream.

```
SURGETANK  ID  DST  DIFFERENTIAL
ELBOTTOM  851.  RTOP 939.     TTOP  944.
RDIAM   13.5  TOIAM  32.     CELERITY  3471.
FRICTION   0.01  AORIFICE   8.4  CDIN  0.7
CDOUT   0.7  FINISH

SURGETANK  ID  CHAM   AIR
ELBOTTOM   340.   ELTOP  393.
CELERITY   3200.   FRICTION   0.02
SHAPE   ELEV  340.     AREA  950.
E   350.  A   1070.  E  360.  A  1122.
E   370.  A   1122.  E  380.  A  1070.
TEMP    70.
PBAR   14.5  N   1.4
WSINIT   353.2
FINISH
```

TJUNCTION

A definition sketch for a Tee Junction is shown below (Figure 17). Only CONDUIT elements may join a TJUNCTION element. Where appropriate, the user may add a dummy conduit between a TJUNCTION and another element type. In this case, the diameter of the connector must be specified as the diameter of the dummy conduit.

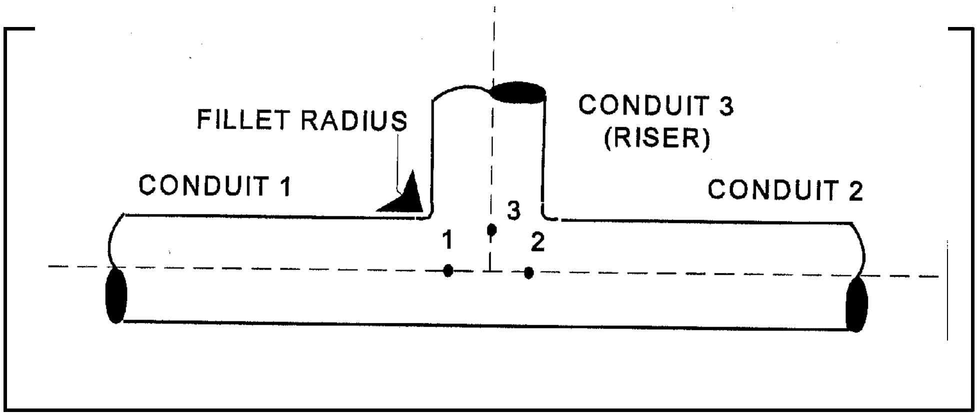


Figure 17. Definition sketch of Tee Junction.

As noted in Chapter 6, Tee Junction losses are computed using formulae developed by Gardel (1965). According to these formulae, with no flow in the riser, the loss of total head from the junction inflow to the riser will be 0.95 times the incoming velocity head. The user has the option of specifying a different value, if desired. This will alter some of the coefficients in the Gardel formulae, affecting computed head losses for other regimes of flow as well.

Secondary Commands	Data	Comments
Identifier		
ID	a	User-defined identifier. This must be a four character alphanumeric word, and must be the first secondary command.
Geometric Data		
FILLET	r	Fillet radius in inches at the riser-conduit junction (Defaults to 0.0).

Special Option

CRISER	r	Specifies the head loss coefficient, in terms of the velocity head associated with inflow to the junction, from the junction inflow to the riser when riser flow is 0. This will affect computed head losses for other flow regimes as well. The default value of 0.95, based on Gardel (1955), should generally be used.

Terminating Command

FINISH

Example: TJUNCTION Command

TJUNCTION ID TJI FINISH

A fillet radius of 0.0 is specified for the above element by default.

TURBINE

The identifier, dimensions, and specifications of each turbine are input using the TURBINE command. Turbine characteristics tables are input separately using the TCHAR command, while the operation of turbines is specified using the OPTURB command.

Secondary Commands	Data	Comments
Identifiers		
ID	a	User defined identifier. This must be the first secondary command.
TYPE	i	i is a user defined number designating which set of characteristics defined under TCHAR corresponds to this turbine. No more than six turbine types may be specified for a system.
Turbine Specifications		
SYNCSPD	r	Synchronous speed in rpm.
FRICTION	r	Friction loss of the turbine-generator unit at synchronous speed in H.P. (Default = 0.0).
WINDAGE	r	Windage loss of the turbine unit at synchronous speed in H.P. (Default = 0.0)
WR2	r	Rotational inertia, WR^2, in lb-ft.^{2}
DIAMETER	r	Diameter of turbine runner throat in feet.
Miscellaneous Command		
AS	a	(see CONDUIT)
FINISH		

Example: TURBINE Command

```
TURBINE   ID    T1       TYPE   1
SYNCSPD     100.      FRICTION   250.    WINDAGE    250.
WR2   148000000.    DIAMETER   20.
FINISH
```

TCHAR

The characteristics of each turbine TYPE are input with a TCHAR command in the form of two tables of either model test data or prototype scale data. The input includes four lists: first, lists of wicket gate position and phi or head which head the columns and rows of the tables then lists of discharge and power or efficiency which form the body of the two tables. These lists must be in the particular order described below.

It is important that the range of characteristics data entered include all operating conditions to be simulated. In particular, care must be taken that the range of phi values is not exceeded due to turbine speed changes. This applies as well when prototype scale data are input, for the values given apply at synchronous speed, and are adjusted according to homologous formulae when speed changes.

There may be one, two, or more turbines in a system of the same TYPE (i.e., for which the same characteristics apply). Up to six sets of turbine characteristics may be input for a single system.

Secondary Commands	Data	Comments
Identifier		
TYPE	i	i corresponds to a TYPE number specified under TURBINE. This must be the first secondary command.
Model Scale Characteristics		
DMODEL	r	Characteristic table values are for a model r inches in diameter. (Defaults to 12 inches.)
GATE	rl, r2, r3...	A list of specified wicket gate positions (in per cent opening) which head the columns of the turbine characteristics tables in either model scale or prototype scale form. There must be at least 3 and no more than 50 values given in ascending or descending order.
PHI	r1, r2,	A list of specified values which head the rows of

r3... the model scale characteristics tables. The term is defined in Section 6.4 by Equation 6.26:

$$\Phi = \frac{\omega D}{2} \;/\; \sqrt{2g\Delta h}$$

where w is in radians/second. Only positive values can be included. There must be at least 3 and no more than 50 values in ascending or descending order.

The GATE and PHI commands must precede the QMODEL and POWER commands.

QMODEL r1, r2, A characteristics table of model scale discharge
QM r3... values (cfs) corresponding to the specified GATE and PHI values. Only positive discharges in the turbine direction can be included. These must be listed in row order (GATE values varied over constant PHI values). There may be up to 625 values.

POWER rl, r2, A characteristics table of model scale power out-
HP r3... put values (H.P.) corresponding to the specified GATE and PHI values. These must be listed in the same order as QMODEL. It is assumed that a Moody-stepup adjustment has been made to these values. There may be up to 625 values.

Prototype Scale Characteristics

GATE r1, r2, Same as for model scale characteristics.
r3...

HEAD rl, r2, For a prototype scale characteristics table, a list
r3... of specified values of head difference across the turbine (feet) will be given in place of PHI values. There must be at least 3 and no more than 50 values.

The GATE and HEAD commands must precede the Q and EFFICIENCY commands.

Q	r1, r2, r3...	A characteristics table of prototype scale discharge values (cfs) corresponding to the specified GATE and HEAD values. Only positive discharges in the turbine direction can be included. These must be listed in row order (GATE values varied over constant HEAD values). There may be up to 625 values.
EFFICIENCY	r1, r2, r3...	A characteristics table of efficiency values (in decimal form) corresponding to the specified GATE and HEAD values. These must be listed in the same order as Q. There may be up to 625 values.

Miscellaneous Command

FINISH

Example: TCHAR Command

The example given for PCHAR illustrating the prescribed order of data input will be repeated in terms of model scale turbine characteristics. Suppose that both the discharge and power tables were as shown:

	GATE				
PHI	1	2	3	4	5
1	1	2	3	4	5
2	6	7	8	9	10
3	11	12	13	14	15

Then the TCHAR command would be as follows:

```
TCHAR    TYPE    1    DMODEL    12.0
GATE    1.    2.    3.   4.    5.    PHI    1.    2.    3.
QMODEL    1.    2.    3.   4.    5.    6.    7.    8.    9.   10.
11.    12.   13.    14.    15.
POWER    1.    2.    3.   4.    5.    6.    7.    8.    9.   10.
11.    12.   13.    14.    15.
```

FINISH

VALVE

To save the user from repeated input of the characteristics of identical valves, each valve is defined to be of a particular type. This type may be GATE, BUTTERFLY, HOWELL, or SPHERICAL, in which case pre-programmed characteristics are invoked (see Section 6.3), or a TYPE number may be specified, the characteristics for this TYPE being input elsewhere with a VCHAR command.

An operating schedule is required of each valve. A particular valve schedule input with a SCHEDULE command is referred to by number.

See PCVALVE and RELIEF for the pressure control valve elements, and ONEWAY for the check valve element.

Secondary Commands	Data	Comments
Identifier		
ID	a	User-defined identifier. This must be the first secondary command.
Characteristics		
GATE		Specifies that the valve type is a disk type gate valve. (0=closed, 100=open)
BUTTERFLY		Specifies that the valve type is a butterfly valve. (90=closed, 0=open)
SPHERICAL		Specifies that the valve type is a spherical valve. (90=closed, 0=open)
HOWELL		Specifies that the valve type is a Howell-Bunger valve. (0=closed, 100=open)
TYPE	i	Specifies that the valve is a special type, with discharge characteristics defined by TYPE i under the VCHAR command.

Geometric Data

DIAMETER	r	Diameter of the valve in feet.

Operating Schedule

VSCHEDULE	i	A user-defined number designating which operating schedule specified under SCHEDULE is to be applied to this valve.

Terminating Command

FINISH		Last command of the input data set.

Examples: VALVE Command

```
VALVE   ID  V1   GATE  DIAMETER   8.0
VSCHEDULE   5  FINISH
```

Often, it may be convenient to input the type and diameter of a valve and the operating schedule number using separate commands.

```
VALVE   ID  V2  TYPE    1   DIAM   12.5  FINISH
VALVE   ID  V2   VSCHED   3   FINISH
```

VCHAR

In defining valve characteristics, either ANGLE or GATEPOS is specified, and either DISCOEF or HDCOEF is specified. The data may be given in either of two ways: (1) by alternating ANGLE or GATE commands with DISCOEF or HDCOEF commands, or (2) by single ANGLE or GATE and DISCOEF or HDCOEF commands, each of which includes the complete list of openings and coefficients.

Separate sets of characteristics may be input for forward and reverse flow within the same VCHAR command.

Each set of characteristics, whether it applies in both directions or in one direction only, must include at least 3 points given in ascending or descending order of valve position. The range of valve positions given must include all valve positions encountered during simulation. No more than 100 points, including both forward and reverse flow characteristics if these are different, may be input for a given valve TYPE.

No more than 6 different valve TYPES (not including BUTTERFLY, GATE, HOWELL, or SPHERICAL) may be input for a single system.

Secondary Commands	Data	Comments

Identifier

Secondary Commands	Data	Comments
TYPE	i	Corresponds to a TYPE number specified under VALVE or RELIEF. There may be several valves with identical properties and therefore the same TYPE Number. This must be the first VCHAR command.

Flow Direction

If separate tables are input for different flow directions, each table must be preceded by a FORWARD or REVERSE command. The positive flow direction for each element is defined with the SYSTEM command (Section 4.2). <u>Neither command should be used if a single table applies in both directions.</u>

Secondary Commands	Data	Comments
FORWARD		Precedes characteristics table input for flow in the assumed positive direction.

REVERSE		Precedes characteristics table input for flow in the negative direction.

Characteristics

ANGLE	r1, r2, r3...	A single value or a list of specified opening angles (in degrees) in a valve characteristics table.
GATEPOS G	r1, r2, r3...	A single value or list of specified gate or valve positions (in percent opening) in a OPENING in a valve characteristics table.
DISCOEF DC	r1, r2, r3...	Asingle value or a list of discharge coefficients corresponding to the specified ANGLE or GATEPOS values. If these are listed, they must be in the same order as the corresponding ANGLE or GATEPOS values. These coefficients represent C_Q, in equation (6.16) of Section 6.3:

$$Q = C_Q \; D^2 \sqrt{g} \; \sqrt{\Delta H}$$

HDCOEF HC	r1, r2, r3...	A single value or a list of head loss coefficients which may be specified as an alternative to DISCOEF. These represent C_H in equation (6.20) of Section 6.3:

$$C_H = \frac{\Delta H}{v^2/2g}$$

Miscellaneous Command

FINISH

Examples: VCHAR Command

VCHAR TYPE 1 ANGLE 0. DISCOEF 1.67
A 45 DC 0.30 A 90 DC 0.08 FINISH

VCHAR TYPE 2

```
FORWARD    OPENING   0.    50.    100.
HDCOEF   1000000.     2.0    0.18             ..
REVERSE   OPENING   0.     25.    50.    75.   100.
HDCOEF   1000000.     10.    3.0    1.0  0.30
FINISH
```

In general, the user will want to input more complete tables than those shown.

4.2 System Command

SYSTEM

The SYSTEM primary command defines the connection of the individual element components, identifies the location of boundary conditions, and specifies the character of system junctions. This input information is used by the program to assemble mathematical equations for the system in a matrix structure.

Preparation of the SYSTEM data is greatly simplified by the construction of a schematic diagram and an element list as described in Section 3.2 and illustrated in Figures 3 and 4. In this diagram, the user identifies system node points, which are the points where elements join. These nodes may be simple connections, or contraction junctions between two conduit elements. Nodes which are junctions of multiple elements involve special considerations further described below. The node numbers defined by the user need not be in any particular order, although it is generally convenient to use an ordered number scheme.

Also identified in the schematic diagram are the presumed flow directions within the system. This establishes a convention for differentiating positive and negative flow in the program output. The assumed positive flow direction in any particular branch of the system is communicated to the program by the order of appearance of the node numbers (upstream to downstream order) which locate the individual elements in the SYSTEM command. The flow direction conventions are listed below:

1. Each element in a branch must have the same assumed flow direction.

2. For PUMP elements, the positive flow direction is in the pumping direction.

3. For TURBINE elements, the positive flow direction is in the generating direction.

4. For PUMP-TURBINE elements, the positive flow direction is in the generating direction.

5. A SURGETANK element is always at the downstream end of a branch.

The secondary commands for the SYSTEM primary command are described below. Each element of a system must be included within the SYSTEM command. This includes simple junctions, which are not defined elsewhere with a separate primary command. The only exceptions are governor elements. Only one SYSTEM command should appear in the input.

Secondary Commands	Data	Comments

Linking Elements

CONDUIT, VALVE, PCVALVE, RELIEF, ONEWAY, PUMP, TURBINE, and PUMP-TURBINE elements link two system nodes. They are located in the system via the ELEMENT secondary commands.

ELEMENT EL	a	Alphanumeric element identifier defined in the various element primary commands.
LINK	i,i	The numerical identifiers of the upstream and downstream nodes, respectively, linked by this element.

Boundary Condition Elements

RESERVOIR, SURGETANK, and FLOWBC elements identify the system boundary conditions. The ELEMENT secondary command also locates these in the system.

ELEMENT EL	a	Alphanumeric element identifier defined in the various element primary commands.
AT	i	The numerical identifier of the node point where the boundary condition applies.

Diameter Change Elements

DCHANGE elements may occur at nodes joining two conduits of different diameters. The ELEMENT secondary command is used to locate these in the system.

ELEMENT EL	a	The alphanumeric element identifier defined in the DCHANGE primary element command.

AT	i	The numerical identifier of the node point where the diameter change is located.

System Junctions

Node points which are the junctions of more than two elements must be identified by either the JUNCTION subcommand or ELEMENT subcommand. Use of the JUNCTION subcommand implies a junction without associated head loss. Mathematically, the program requires only a continuity equation to model a junction of this type. As many as six branches may be joined to JUNCTION nodes.

JUNCTION		Indicates a simple junction without head losses.
AT	i	The numerical identifier of the node point where the simple junction is located.

If head losses are to be considered in the junction, the properties must be defined via the TJUNCTION primary element command (see last section). This TJUNCTION element is then identified with its node location by the ELEMENT subcommand below. Mathematically, such a node is represented with both continuity and momentum equations.

ELEMENT EL	a	Alphanumeric element identifier defined in the TJUNCTION primary element command.
AT	i	The numerical identifier of the node point where the TJUNCTION element is located.
RISER	i	The numerical identifier of the node point to which the riser pipe of the TJUNCTION is connected.

Specifying Node Elevations

It is not generally required to specify node point elevations. However, if pressure head (piezometric head minus elevation) is requested output for any node then the elevation of that node must be given as part of the SYSTEM command. Also, node elevations must be given at points where potential column separation is to be flagged. Pairs of NODE and ELEV subcommands are used for this purpose.

NODE	i	The numerical identifier of the node point whose elevation is to be specified.
ELEV	r	Elevation of the node point in feet.

Miscellaneous Command

FINISH	The final command of the SYSTEM input stream.

Example: SYSTEM Command

Refer to Figure 3 and Table 1, which show a schematic diagram and an element list for a hypothetical system. Based on these, the SYSTEM command would be as follows:

```
SYSTEM
EL   HW    AT     1
EL   C1    LINK   1   2
EL   C2    LINK   2   3
EL   DC1   AT     3
EL   C3    LINK   3   4
JUNCTION   AT     4
EL   DUM   LINK   4   5
EL   FBC   AT     5
EL   C4    LINK   4   6
EL   V1    LINK   6   7
EL   C5    LINK   7   8
EL   TJ    AT     8
EL   C6    LINK   8   9
EL   T1    LINK   9   10
EL   TW    AT     10
EL   C7    LINK   8   11
EL   DC2   AT     11
EL   ST    AT     11
NODE  9   ELEV   121.7
NODE  3   ELEV   147.3
FINISH
```

4.3 Output Commands

Program output is controlled by the DISPLAY, ECHO/NOECHO, HISTORY, PLOTFILE, SPREADSHEET, SNAPSHOT, and TEXT commands. DISPLAY, HISTORY, and SNAPSHOT identify the desired printed output, while the PLOTFILE command specifies data to be stored in a disk file for later plotting by a graphics program. The SPREADSHEET command specifies time history data to be stored in a file formatted for easier import by spreadsheet programs. ECHO and NOECHO are used to control the echo printing of the program input data. Also, a text message to be printed with the output tables (in addition to the simulation "title" entered in the first line of input) can be entered using the TEXT command.

The number of lines per page and the number of decimal places to be printed in output tables can be specified with the HISTORY command.

The number of decimal places to be printed can also be specified with the SPREADSHEET command.

DISPLAY

The DISPLAY command controls the printing of output tables which display the parameters specified as input data. These tables are useful to check that the desired data have been properly input, but generally are not necessary for repeated runs of the same system. The DISPLAY command identifies those tables which are desired and those not desired.

Secondary Commands	Data	Comments
ALL		Turns all of the display options on.
OFF		Turns all of the display options off.
STANDARD		Turns on all display options except the machine and valve characteristics (Default).
CONDUIT		This command will print a table displaying the conduit input data.
MACHINE		This command will print a table displaying the input data for pumps, pump-turbines and turbines. In addition, it may be followed by the following modifiers to display the tabular turbo-machine characteristics:
PUMP		Displays the tables of head and torque versus speed and discharge for each pump type.
P-T		Displays the head and torque tables for each gate opening for the pump-turbine.
TURBINE		Displays the tables of homologous turbine data.
VALVE		This command will print a table of the valve input data. In addition, it may be followed by the following modifier:
CHARACTERISTICS		Displays the tables of coefficient of discharge versus valve opening for each valve type.

TANK	Displays the table of surge tank input data.
BC	Displays the table of boundary condition input data.
SCHEDULES	Displays tables of the boundary condition, valve, and load control schedules.
SYSTEM	Displays a schematic of the system connectivity.
OUTPUT	Displays a table of the output history and plotfile requests.
FINISH	Ends the DISPLAY command.

ECHO/NOECHO

The ECHO and NOECHO commands control the program printback of the input data. The commands consist of single primary commands and are not followed by the FINISH command. The commands are:

ECHO	This command causes the program to reprint all subsequent input data in its entirety. This is the default value and is generally necessary only if the NOECHO command is used previously.
NOECHO	This command suppresses echo printing of subsequent input commands.

HISTORY

To specify that a time history of a certain element should be printed as output, the identifier of that element must be listed under a HISTORY primary command, followed by the secondary command(s) which indicates the variable(s) of interest. In the same manner, the heads and discharges at any node point may be specified as output.

Optional format specifications for tabular output may also be entered.

Secondary Commands	Data	Comments
Node Identification		
Location		
NODE	i	The identifier of the node point for which an output time history is desired.
UPSTREAM DOWNSTREAM		For nodes associated with multiple head values (e.g., diameter changes) or discharge values (e.g., surge tanks with outlets) these qualifiers specify which of the possible values are to be output. In the absence of these commands, the upstream value will be output.
TONODE	i	When a junction node is specified, the particular branch for which variable values are desired is specified by following the NODE sub-command with a TONODE sub-command which indicates the node number which is part of that branch and adjacent to the junction node.
Variables		
Q		Discharge at the specified node in cfs.
HEAD		Total head in terms of the energy gradient elevation in feet at the specified node.

PIEZHEAD		Piezometric head in terms of the piezometric elevation in feet at the specified node.
PRESSURE		Pressure head (piezometric head minus node elevation) at the specified node. When this variable is requested output for a node, the node point elevation must be supplied in the SYSTEM command.
GPM		Discharge at the specified node in gallons per minute.
PSI		Pressure head at the specified node in pounds per square inch.

Element Identification

Location

ELEMENT	a	Alphanumeric identifier of the element at which the history of a particular parameter is desired.
ATNODE	i	The upstream node point of the element of interest. This additional specification is necessary only when an element appears more than once in the system. (This must follow the element command!)

Parameters

Q		Discharge at upstream end of element.
HEAD		Head difference across a machine element.
SPEED		Rotational speed of turbomachine.
POSITION		Wicket gate or valve position.
ELEV		Water surface elevation (in surge tank or riser of differential surge tank).

ELTANK		Water surface elevation of the outer tank of a differential surge tank.
POWER		Power.
TORQUE		Torque.
OUTFLOW		Flow which leaves the system from a surge tank side outlet.
VOLUME		Air volume in air chamber surge tank.
PRESSURE		Air pressure in an air chamber surge tank.
TEMP		Air temperature in an air chamber surge tank.
QAIR		Air mass inflow (positive value) or outflow rate for a vented air chamber surge tank.
AMASS		Air mass in an air chamber surge tank.

Format

LINES	i	Number of lines per page desired in printed output. (Defaults to 50.)
DECIMAL	i	Number of decimal places displayed in the output. This will apply to all values printed in the time history output table. The maximum is 3 and the minimum is 0. (Defaults to 1.)

Terminating Command

FINISH

Example: HISTORY Command

```
HISTORY  NODE  6   PIEZ  Q   FINISH
HISTORY  ELEM   T1  SPEED
NODE   3 TONODE   8 Q
```

```
        LINES 60    DECIMAL 2
FINISH
```

The first request will cause the printing of the piezometric head and discharge versus time at node number 6. The second will cause the rotational speed of turbine T1 to be output as a function of time, and the third request will cause the discharge at junction node 3 in the branch leading from node 3 to node 8 (where node 8 is the next adjacent node in that branch) to be output.

SPREADSHEET

The secondary commands associated with SPREADSHEET are exactly the same as those of HISTORY. The SPREADSHEET command causes the data to be written to a file formatted for easier import by spreadsheet programs. This is an ASCII file which includes only the time history output tables without a summary table or other extra text. Column headers are written only once, there are no page breaks, and output is written to as many columns as necessary without "wrapping." Each column header looks like a single word without spaces, for example "NODE_NO_80."

PLOTFILE

The secondary commands associated with PLOTFILE are exactly the same as those of HISTORY. PLOTFILE causes data to be written to a file for subsequent plotting using the graphics program, while HISTORY causes data to be printed as output.

SNAPSHOT

With the use of the SNAPSHOT command the user may obtain printed output showing the values of head and discharge at all points throughout the system at a specified time — obtaining, in a sense, an instantaneous hydraulic picture of the system.

Secondary Command	Data	Comments
TIME	r	Time at which SNAPSHOT output is desired. Up to 10 separate times may be requested for a single simulation.
FINISH		

Example: SNAPSHOT Command

SNAPSHOT TIME 0.0 10.0 20.0 FINISH

TEXT

Text entered in the following two lines will be printed with the headings of the simulation summary and output tables (in addition to the simulation "title" entered on the first line of input). Text entered on the same line as the TEXT subcommand is ignored. Any commands entered on the two lines following TEXT will not be processed. If desired text is only one line, leave the second line blank. No FINISH command is necessary. Up to 80 characters may be entered per text line.

Example: TEXT Command

TEXT
Load Rejection Simulation Number 3
30 Second Wicket Gate Closure Time

4.4 Simulation Control Commands

The simulation control commands define the computational parameters, the operation of hydraulic machinery, and schedule of valve and gate opening, generator load, and boundary conditions.

CLOSURE

This command is an adjunct to the SCHEDULE command. When specifying wicket gate operation, using SCHEDULE, a special option may be selected which defines a linear, constant rate closure with "cushioning" (i.e., reduced rate of gate movement) at final closure. The CLOSURE command is used to specify a dimensionless curve representing a final, non-linear "cushioned" portion of the gate closure. The linear gate closure rate and the dimensions of this non-linear curve are input separately under SCHEDULE.

This command must be entered only if the special gate closure option is invoked under SCHEDULE and the CURVE subcommand is given there.

Secondary Commands Data Comments

Pairs of dimensionless time and gate position points are entered. Up to 20 pairs may be entered. Figure 42 depicts TF and GF.

Secondary Commands	Data	Comments
TFRAC TF	r	Fraction of time from the beginning of the cushioning curve to complete closure. The first value must be 0.
GFRAC GF	r	Fraction of the gate position at which the cushioning curve begins. The first value must be 1.
FINISH		

Example: CLOSURE Command

CLOSURE TFRAC 0. GFRAC 1. TF 0.25 GF 0.6
TF 0.5 GF 0.3 TF O.75 GF 0.1 TF 1.0 GF 0.0
FINISH

CONTROL

The computational and output time steps may have different values for different periods of the simulation. This requires repeated groups of DTCOMP, DTOUT, and TMAX subcommands under the same CONTROL primary command. The computational time step will typically range from about 0.05 seconds to 5 seconds (see Sections 3.2 and 3.6).

Secondary Commands	Data	Comments
THETA	r	User-specified weighting ratio to be used in the finite difference numerical solution of the differential equations of flow in the system. This is a number from 0 to 1 which will usually range from 0.4 to 0.6. The default value is 0.6.
DTCOMP	r	Computational time step in seconds. Up to 10 different values may be specified.
DTOUT	r	Time interval (in seconds) at which the output parameters are printed or stored. Up to 10 different values may be specified.
TMAX	r	Last value of time (in seconds) for which DTCOMP and DTOUT apply. The last TMAX value which appears defines the time at which the simulation terminates.

Minimum Computational Time Step

When RELIEF or ONEWAY elements are simulated, computations may sometimes be performed at time steps shorter than specified with DTCOMP. If, as a result, simulation time becomes excessive or matrix solution problems arise, the user may want to specify a minimum time step.

DTMIN	r	Minimum computational time step in seconds. Defaults to 0.001.

Accuracy Test Control Commands

The WHAMO computational routines include checks for numerical errors. These can sometimes result from the round off necessary to store a real number in a finite digital memory space. In particular, the matrix solver can be subject to numerical inaccuracies if the matrix is near singular.

If such an inaccuracy is detected, a warning message is printed with the output. If the user desires, however, he may either suppress the accuracy check or he may request printing of the results for each check at each time step. He may, additionally, adjust the tolerance of the error tests.

This is accomplished by including the ACCUTEST subcommand under CONTROL, followed by any of the options described below.

ACCUTEST	FULL	Specifies full printout during simulation.
	ON	Specifies printout if error is detected (Default condition).
	OFF	No testing.
HTEST	r	Tolerance (in feet) in a test which compares, at certain points in the system, the head values computed by matrix solution and by back substitution (Defaults to 0.1 ft).
QTEST	r	Tolerance (in cfs) in a similar test comparing computed discharges (Defaults to 1.0 cfs).
EPSILON	r	Relative tolerance in the S.S.P. matrix solver routine DGELG (IBM, 1970). The default value of 10^{-14} is as recommended in the SSP manual.

Miscellaneous Command

FINISH

Examples: CONTROL Command

CONTROL DTCOMP 0.1 DTOUT 0.5 TMAX 10.0 FINISH

```
CONTROL   DTCOMP  0.1   DTOUT  0.5   TMAX   10.0
   DTCOMP  0.5   DTOUT  0.5   TMAX   15.0
   DTCOMP  1.0   DTOUT  1.0   TMAX   30.0
   DTCOMP  5.0   DTOUT  5.0   TMAX  120.0   FINISH
```

Note that THETA, DTMIN, and the accuracy test control commands do not normally need to be specified.

FLUID

The FLUID command is used to specify the fluid density when simulating fluids which have a density different from that of water. Other fluid properties such as viscosity and compressibility may affect the values of friction factor and celerity which the user specifies using the CONDUIT and SURGETANK commands.

Secondary Commands	Data	Comments
DENSITY	r	Fluid density in pounds per cubic foot. The default value is 62.4.
FINISH		

INITIAL

This command is not normally required input. The initial steady state of a system is automatically computed as part of every simulation. However, if the steady state computation does not converge to a solution, it will be necessary for the user to input estimates of initial conditions. In most cases, steady state convergence can be achieved by supplying an estimate of the initial discharge for some or all of the serial branches of the system. This is done with pairs of BRANCH and QEST commands.

If steady state convergence cannot be achieved, even with the use of the INITIAL command, it may be necessary to begin the simulation with no flow by closing wicket gates or a valve, then slowly bring the system to the desired initial state during the transient simulation.

Secondary Commands	Data	Comments
Branch Estimates		
BRANCH B	i	Serial branch number. The WHAMO output will display each branch of a system and show its designation number. This command must precede each QEST command.
QEST Q	r	Estimated initial steady state discharge (cfs) the branch designated in the preceding BRANCH command.

For simulations involving a turbine or pump-turbine generating under governor control, the program must hunt for the wicket gate position which produces the specified power output. If a successful steady state solution is not reached, the user should input an estimate of initial turbine wicket gate position and differential head.

Turbomachine Estimates

ELEMENT	a	Identifier of the turbo-machine for which an estimate is being made. This must precede the GATE and HEAD command.

GATE	r	Estimated (or pre-computed) initial wicket gate position.
HEAD	r	Estimated (or pre-computed) initial head difference across the turbomachine.

If steady state solution does not converge and there are one or more PCVALVE or RELIEF elements in the system, it may be necessary to specify the initial valve position. Normally, PCVALVE or RELIEF valve position is computed as a function of pressure according to a user defined relation. At the start of steady state computations, however, pressures are unknown so the program automatically assigns an initial estimate of PCVALVE or RELIEF valve position. In some cases, errors in initial estimates may lead to diverging oscillations of pressure and valve positions during steady state computations which preclude a stable solution. The user may then specify an initial PCVALVE or RELIEF valve position with an INITIAL command.

RELIEF Element Specification

ELEMENT	a	Identifier of the PCVALVE or RELIEF element for which the specification is being made. This must precede the OPEN command.
OPENING ANGLE GATE	r	Initial valve position.

Miscellaneous Command

FINISH

Examples: INITIAL Command

```
INITIAL    BRANCH   1    QEST    3460.
B  2   Q  484.  B  5  Q  800.
FINISH
```

Initial estimates of steady state discharge in 3 branches of a system are supplied with the INITIAL command above. Often, an accurate discharge estimate for one main stem branch of a system is sufficient to insure convergence in those cases where an INITIAL command is required.

In the example below, an initial head difference and gate position is estimated across machine element P1.

```
INITIAL   ELEMENT   P1   HEAD   120.   GATE   88.   FINISH
```

OPPUMP

An operation specification must be supplied for each pump system. Only one of the operational schemes below may be chosen for each unit. It will usually be more convenient and less confusing to include separate OPPUMP commands for each pump element.

Secondary Commands	Data	Comments
Identifier		
ID	a	Identifier of the pump whose operation is being specified. This must be the first operation specification command.
Operation Specification		
PUMP		Specifies constant speed operation of the pump.
SHUTOFF		Specifies constant speed operation until shutoff at time TOFF.
TOFF	r	Time of pump shutoff in seconds. (Default to 0.0.)
OFF		Pump not operated, i.e. has no impact on head and flow computations in the simulation.
Miscellaneous Command		
FINISH		

Example: OPPUMP Command

OPPUMP ID P1 SHUTOFF FINISH

A shutoff time of 0.0 seconds has been specified by default.

OPPT

An operation specification is required of each pump-turbine in the system. Only one of the operational schemes below may be chosen for each unit. It will usually be more convenient and less confusing to include separate OPPT commands for each pump-turbine element.

Unless the OFF mode is specified, a schedule of wicket gate position or governor load is required of each pump-turbine. This schedule is input separately with a SCHEDULE command and referred to by number under OPPT.

Secondary Commands	Data	Comments
Identifier		
ID	a	Identifier of the pump-turbine whose operation is being specified. This must precede the operation and schedule specification commands.
Operation Specification		
PUMP		Specifies synchronous speed operation in the pump mode.
SHUTOFF		Specifies synchronous speed pump operation until shutoff at time TOFF.
GENERATE		Specifies synchronous speed operation in the turbine mode.
REJECT		Specifies synchronous speed turbine operation until load rejection at time TOFF.
TOFF	r	Time of shutoff or load-rejection in seconds. (Default to 0.0.)
GOVERN		Specifies governor controlled operation in the turbine mode serving an isolated load. (Turbine speed is allowed to vary.)

STARTUP		Specifies turbine startup from 0 rpm under non-governor wicket gate control. Turbine speed is allowed to vary, with no transition to synchronized operation. (Startup in pumping mode is not simulated.)
OFF		Pump-turbine not operated. Wicket gates closed.

Schedule Designation

VSCHEDULE	i	Designates the operating schedule specified under SCHEDULE to be applied to the wicket gates of this pump-turbine. This is required input if the PUMP, SHUTOFF, GENERATE, REJECT, or STARTUP mode is selected.
LSCHEDULE	i	Designates the schedule of generator load applied to the pump-turbine specified separately under SCHEDULE. This is required input if the GOVERN mode is selected.

Note that a gate schedule and a load schedule cannot both be specified for the same unit in the same simulation.

Miscellaneous Command

FINISH

Examples: OPPT Command

```
OPPT   ID   PT1   SHUTOFF   VSCHEDULE   5
FINISH
```

A time of 0.0 has been specified above for the pump shutoff by default.

```
OPPT   ID  PT2   GOVERN   LSCHED   2   FINISH
```

OPTURB

An operation specification is required of each turbine in the system. Only one of the operational schemes below may be chosen for each unit. It will usually be more convenient and less confusing to include OPTURB commands for each turbine element.

Unless the OFF mode is specified, a schedule of wicket gate position or governor load is required of each turbine. This schedule is input separately with a SCHEDULE command and referred to by number under OPTURB.

Secondary Commands	Data	Comments
Identifier		
ID	a	Identifier of the turbine whose operation is being specified. This must precede the operation and schedule specification commands.
Operation Specification		
GENERATE		Specifies synchronous speed operation of the turbine.
REJECT		Specifies synchronous speed operation until load rejection at time TOFF.
TOFF	r	Time of load rejection in seconds. (Default of 0.0.)
GOVERN		Specifies governor controlled turbine operation serving an isolated load. (Turbine speed is allowed to vary.)
STARTUP		Specifies turbine startup from 0 rpm under non-governor wicket gate control. Turbine speed is allowed to vary, with no transition to synchronized operation.
OFF		Turbine not operated. Wicket gates closed.

Schedule Designation

VSCHEDULE	i	Designates the operating schedule specified under SCHEDULE to be applied to the wicket gates of this turbine. This is required input if the GENERATE, REJECT, or STARTUP mode is selected.
LSCHEDULE	i	Designates the schedule of generator load applied to the turbine specified separately under SCHEDULE. This is required input if the GOVERN mode is selected.

Note that a gate schedule and a load schedule cannot both be specified for the same unit in the same simulation.

Miscellaneous Command

FINISH

Examples: OPTURB Command

```
OPTURB   ID   T1   REJECT   VSCHEDULE   4
FINISH
```

A time of 0.0 has been specified above for the load rejection by default.

```
OPTURB   ID   T2   GOVERN   LSCHED   1   FINISH
```

SCHEDULE

Operating schedules of valves as well as schedules of generator load, wicket gate opening, reservoir elevation, and flow boundary discharges are specified with the SCHEDULE command. A single valve schedule may apply to more than one valve, a single load schedule to more than one turbine, and elevation or discharge schedules may apply to multiple boundary conditions.

Up to 25 points may be included in a single schedule. If the simulation time exceeds the last specified time value, the operating value will be held constant for the remainder of the simulation at the last specified value in the schedule.

It is important to specify a sufficient number of points to completely define the curve. Often, a number of points will be needed even for linear portions of the curve, due to the nature of the cubic interpolation routine. It is recommended that the schedule variable be plotted after simulation to check the interpolation.

A number of schedules may be included in a single SCHEDULE command, or each schedule may be given in separate SCHEDULE commands. A total of 15 schedules may be specified for a simulation.

Secondary Commands	Data	Comments

Valve or Gate Schedule

Identifier

VSCHEDULE GSCHEDULE	i	Corresponds to a VSCHEDULE or GSCHEDULE number specified under VALVE, OPTURB, or OPPT. This identifier must precede the schedule definition commands. (Note: VSCH and GSCH are synonymous).

Schedule Definition

The valve (or wicket gate) opening as a function of time is defined through a series of time versus opening data pairs. The data may be given by alternating the TIME and ANGLE or GATEPOS commands, or by a DELT command and a single ANGLE or GATEPOS command followed by a list of specified values.

TIME T	r	The time value of the data pair (seconds). The first value must be 0.0 with subsequent values in increasing order.
ANGLE A	r	The valve opening angle (degrees) at the time specified. Fully open is 0 degrees; Shut is 90 degrees. Applies to butterfly, spherical and user-defined valve types.
GATEPOS G OPENING	r	The valve or gate opening position percent of fully open at the time specified. Applies to wicket gates and gate, Howell-Bunger and user-defined valve types.
DELT	r	When a valve history is given in constant time intervals beginning at 0.0, the user may specify that interval instead of listing each time value.

If desired, a sinusoidal variation may be superimposed on the specified valve schedule using the following commands.

AMPLITUDE	r	Amplitude of superimposed sinusoidal variation (degrees or percent) (Defaults to 0).
FREQUENCY	r	Frequency of superimposed sinusoidal variation (1/second).
PHASE	r	Initial phase angle in degrees (Defaults to 0).
DURATION	r1, r2	Beginning and ending time, respectively, during which the superimposed sinusoidal variation applies (seconds). If this command is not entered, any specified variation will apply for the entire simulation.

Special Gate Closure Option

Instead of, or in addition to, entering a table of time and gate position, the user may call upon a standard gate closure curve. The standard gate closure includes 4 parts: an initial starting time for the gate to accelerate to a specified closing rate, straight line closure at the specified rate, a decelerating time during which the closure rate is reduced to the slower, cushioned rate, and finally straight line closure at the cushioned rate.

Alternatively, a non-linear final closure may be specified instead of a decelerating time and final linear closure rate. In this case, the user must specify a dimensionless final closure curve elsewhere with a CLOSURE command. The dimensions of this curve (i.e., the gate position at which it begins and the time duration from beginning of the curve to complete closure) must be specified here.

The initial gate position at time 0 must be input with TIME and GATE sub-commands. Standard closure need not start at time 0. It will start at the last time for which the gate position is explicitly given.

Details of the standard gate closure are given in Section 6.8.

TACCEL	r	Time of acceleration in seconds from stationary gate to closure at the specified rate. Defaults to 0.25.
RTCLOSE	r	Rate of closure in terms of seconds for full stroke. Sometimes defined as twice the elapsed time from 75 percent gate to 25 percent gate.
GCUSHION	r	Gate position (percent opening) at which gate deceleration or the non-linear closure curve begins. Normally, this will occur at or slightly below speed-no-load gate. Typical values are 20 percent for Kaplan turbines and 10 percent for Francis turbines. Defaults to 10 percent.

The TDECEL and RTCUSH commands below apply only for linear final closure.

TDECEL	r	Time of deceleration in seconds from rapid gate closure to cushioned gate closure. Defaults to 1.0.
RTCUSH	r	Rate of cushioned final closure in terms of seconds for full stroke. Defaults to 150.

The CURVE and TCUSH commands below apply only for non-linear final closure.

CURVE		Indicates that final, cushioned closure is defined by a dimensionless curve specified with a CLOSURE command.
TCUSHION	r	Time in seconds from the beginning of the cushioned final closure curve to complete closure. Typical values are 25 seconds for Kaplan turbines and 15 seconds for Francis turbines. Defaults to 15 seconds.

Generator Load Schedule

Identifier

LSCHEDULE	i	Corresponds to a LSCHEDULE number specified under OPTURB or OPPT. This identifier must precede the schedule definition commands.

Schedule Definition

The generator load as a function of time is defined through a series of time versus load data pairs. The data may be given by alternating TIME and LOAD commands, or by a DELT command and a single LOAD command in a manner similar to the valve schedule.

TIME T	r	
DELT	r	
LOAD L	r	The generator power demand which must be supplied by the turbine given in (HP), at the time specified.

If desired, a sinusoidal variation may be superimposed on the specified load schedule using the following commands described for valve schedules.

AMPLITUDE	r	Amplitude of superimposed sinusoidal variation (HP). Default to 0.
FREQUENCY	r	

PHASE	r	
DURATION	rl, r2	

Time Varying Boundary Conditions

Identifiers

HSCHEDULE	i	Corresponds to an HSCHEDULE number specified under RESERVOIR.
QSCHEDULE	i	Corresponds to a QSCHEDULE number specified under FLOWBC.

The identifier must be given before the schedule definition commands.

Schedule Definition

The discharge or head as a function of time is defined through a series of time versus discharge or head data pairs. The data may be given by alternating TIME and Q or HEAD commands, or by a DELT command and a single Q or HEAD command, in a manner similar to the valve schedule.

TIME T	r	
DELT	r	
ELEV H	r	The elevation or head (ft.) at a reservoir boundary at the time specified.
Q	r	The discharge (cfs) at a flow boundary at the time specified.

If desired, a sinusoidal variation may be superimposed on the specified boundary condition schedule using the following commands described for valve schedules.

AMPLITUDE	r	Amplitude of superimposed sinusoidal variation (ft. or cfs). Defaults to 0.

FREQUENCY r

PHASE r

DURATION r1, r2

Terminating Command

FINISH

Examples: SCHEDULE Command

A valve closure history from an opening of 90 degrees to 0 degrees in sixty seconds is specified in the first command group below.

```
SCHEDULE    VSCHEDULE 1         TIME    0.0   ANGLE    90.0
T 10.0 A    73.0 T   20.0  A  57.0
T 30.0 A    40.0 T   40.0  A  23.0
T 50.0 A     7.0 T   55.0  A   2.0
T 58.0 A     1.0 T   59.0  A   0.4
T 60.0 A    0.0
```

An alternative to specifying each data pair is illustrated below for a boundary discharge schedule.

```
SCHEDULE    QSCHED   1     DELT   1.0
0   200.0    150.0    100.0    50.0    0.0
FINISH
```

The final example includes the special option gate closure plus a specified constant load with a superimposed sinusoidal variation. (It is unlikely that both would be invoked in the same simulation.)

```
SCHEDULE    GSCHEDULE    2     TIME     0.     GATE     78.5
TACCEL     0.8     RATE      8.    GCUSHION     20.
TDECEL     0.5     RTCUSH     100.
LSCHEDULE    1     TIME     0.     LOAD     43260.
AMPLITUDE    1500.    FREOUENCY     0.12
FINISH
```

SEPARATION

A special message is printed warning of potential column separation at node points for which elevations have been input (see SYSTEM command) if computed pressure drops below the critical value. The critical pressure head is set at -20 feet (piezometric head 20 feet below specified node elevation) unless the user modifies it using this command. Potential column separation will not be flagged at nodes for which elevations have not been specified.

Secondary Commands	Data	Comments
PRESSURE	r	If the computed pressure head (piezometric head minus node elevation) is less than r at any node with elevation specified a warning message is printed. The default value is -20 feet. (Note that this is gage pressure so a negative value denotes less than atmospheric pressure.)
FINISH		

4.5 Execution Control Commands

A number of commands are used to control program execution. These are single word commands which do not require a FINISH command.

The CHECK and IONLY commands are used to limit computations.

CHECK	Indicates that the run is a 'check' run and should not proceed with the transient simulation or calculation of initial conditions. Network data will be processed and checked. The command can be placed anywhere prior to a GO command.
IONLY	Indicates that only the initial conditions should be generated, without subsequent simulation of the transient conditions. This command can be placed anywhere prior to a GO command.

The GO command must follow all data and simulation specification commands.

GO	Indicates that the end of the data defining the system has been reached and that execution should begin.

Simulation computations initiated by the GO command are carried out to completion, including output. It is possible to have any number of simulations performed as part of a single job submission. If this is desired, the GO command should be immediately followed by a RERUN or NEWRUN command, which in turn will be followed by a title line, a set of data and simulation specification commands, and another GO command to initiate the new simulation.

RERUN	Placed after the GO command to indicate that the previous simulation should be repeated with certain modifications. This command must be followed by a new title line and then commands to modify the input data given for the previous simulation. If not specifically altered with new commands, data and specifications from the previous simulation will apply to the new simulation.

NEWRUN	Place after the GO command to indicate that an entirely new set of input data for a new simulation follows. Input data and specifications from the previous simulation do not apply to the new simulation.

The final entry must always be GOODBYE, which indicates that there is no subsequent simulation to be performed.

GOODBYE	Terminates program execution.

5 Example Applications

Examples of input data for some hydraulic systems are presented in this section. Please note that this input data was prepared with the command file method using a text editor.

5.1 Valve Closure in a Simple Pipeline

This example is taken from Parmakian (1963). A schematic diagram for this system is shown in Figure 18 and an element list in Table 3. Both of these are as described in Section 3.2. The input file is shown in Figure 19.

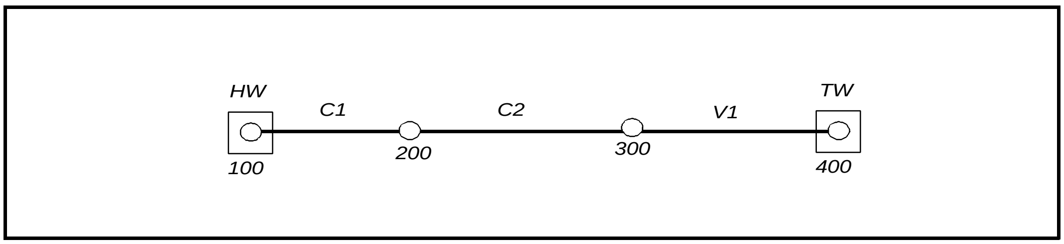


Figure 18. Schematic diagram of simple pipeline system.

Table 3. Element list for simple pipeline system.

Element I.D.	Type	Node Location	U.S. Node	D.S. Node
HW	Head Boundary	100	—	—
C1	Conduit	—	100	200
C2	Conduit	—	200	300
V1	Valve	—	300	400
TW	Head Boundary	400	—	—

The valve characteristics shown in the input are artificial. They were synthesized, along with the valve schedule, to duplicate the gate closure time relation shown in

```
   VALVE CLOSURE IN A SIMPLE PIPELINE

   C THE TITLE CARD ABOVE MUST BE THE FIRST DATA CARD
   C SYSTEM CONNECTIVITY IS SPECIFIED BELOW WITH THE "SYSTEM" COMMAND

   SYSTEM
    EL HW AT 100
    EL C1 LINK 100 200
    EL C2 LINK 200 300
    EL V1 LINK 300   400
    EL TW AT 400
    FINISH

C   ELEMENT PROPERTIES

   RESERVOIR ID HW ELEV 500. FINISH
   CONDUIT ID C1 DIAM l0. CELERITY 300. FRICTION 0.0000l LENGTH 1500.
   NUMSEG 5 FINISH
   CONDUIT I3 C2 AS C1 FINISH
   VALVE ID Vl DIAM 10. TYPE 1 FINISH
   RESERVOIR ID TW ELEV 0. FINISH

   C  VALVE
   CHARACTERISTICS

   VCHARACTERISTICS TYPE 1
    GATE 0. 10. 20. 40. 60. 80. 100.
    DISCOEF 0.  0.0067 0.0200 0.0332 O.0465 0.0598 0.0664
    FINISH

   C  VALVE OPERAT ION

   VALVE ID V1 VSCHED 1 FINISH
   SCHEDULE VSCHED 1
   DELT 1.0 GATE 100. 80. 60. 40. 20. 10. n.
   FINISH .

   C OUTPUT REQUESTS

   HISTORY
    NODE 200 HEAD
    NODE 300 HEAD
    FINISH
    DISPLAY ALL FINISH

C COMPUTATIONAL PARAMETERS

   CONTROL DTCOMP 0.1 DTOUT 0.5 TMAX 14.5 FINISH

C EXECUTION CONTROL

   CHECK
   GO
   GOODBYE
```

Figure 19. Input for valve closure in a simple pipeline.

Figure 24 of Parmakian. The important point is to show how valve characteristics, no matter how determined, are input to the model.

Note that DISPLAY ALL and CHECK commands are included with the input. The former causes all available (and relevant) input data display tables to be printed, while the latter aborts execution of the simulation. After checking the input data, these commands can be removed and a simulation performed.

The pipe was divided into two conduit elements in order to get output at the midpoint.

5.2 Simple Pipeline With a Surge Tank

This example also comes from Parmakian (1963). It is taken from Section 68 beginning on page 124.

Simple use of a flow boundary element and dummy conduit elements is illustrated. Figure 20 shows the system, Table 4 shows the element list, and Figure 21 shows the program input. The flow boundary represents a gate closure while the dummy conduits are needed to prevent "one element branches" which result when boundary elements are joined to junction elements. As in the previous example, a very low conduit friction is used to approximate an ideal frictionless pipeline. High values of celerity are used to reduce the effect of water hammer on the surge.

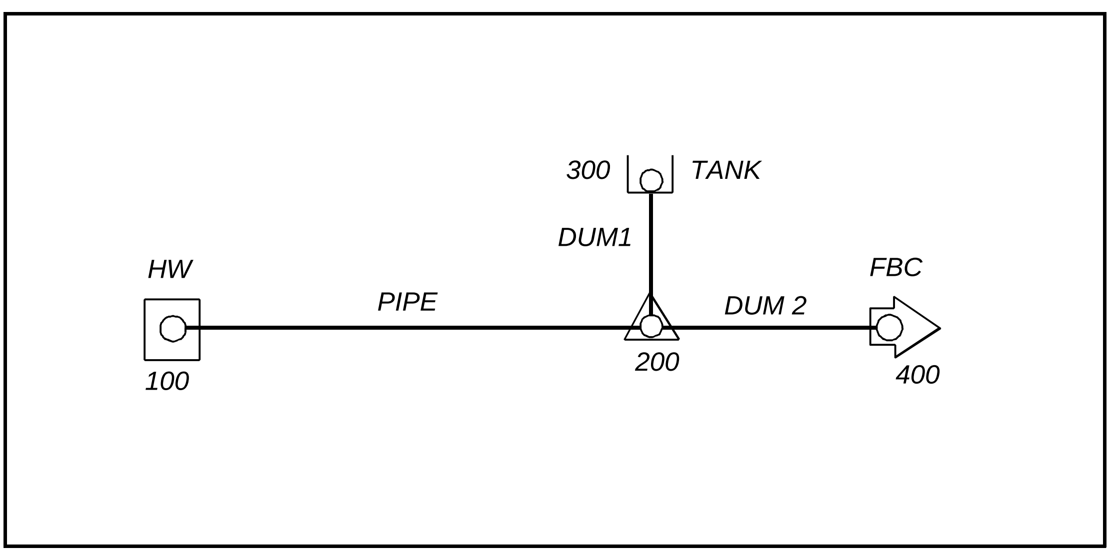


Figure 20. Simple pipeline with surge tank.

```
SIMPLE PIPELINE WITH A SURGE TANK

    C   SYSTEM CONNECTIVITY

    SYSTEM
     EL HW AT 100
     EL PIPE LINK 100 200
     JUNCTION 4T 200
     EL DUM1 LINK 200 300
     EL TANK 4T 300
     EL DUM2 LINK 200 400
     EL FBC AT 400
     FINISH

    C ELEMENT PROPERTIES

    RESERVOIR ID HW ELEV 500. FINISH
    CONDUIT ID PIPE LENGTH 3000. NUMSEG 10 DIAM 10. FRICTION 0.00001
             CELERITY 6000. FINISH
    CONDUIT ID DUM1 DUMMY FINISH
    SURGETANK ID TANK DIAM 20. ELTOP 600. ELBOTTOM 450. FRICTION 0.00001
             CELERITY 6000. FINISH
    CONDUIT ID DUM2 DUMMY FINISH
    FLOWBC ID FBC QSCHED 1 FINISH

   C   FLOW BOUNDARY OPERATING SCHEDULE

    SCHEDULE QSCHED 1
     TIME O. Q 843. T 0.01 Q 0.0
     FINISH

    C OUTPUT REQUESTS

    HISTORY
      ELEM TANK ELEV
      NODE 300 0
      FI N ISH
    DISPLAY ALL FINI SH
    SNAPSHOT TIME 30.3 FINISH

    C COMPUTATIONAL PARAMETER

    CONTROL
     DTCOMP 0.1 DTOUT 5.0 TMAX 10.0
     DTCOMP 1.0 DTOUT 5.0 TMAX 25.0
     DTCOMP 1.0 DTOUT 1.0 TMAX 35.0
     FINISH

    C EXECUT ION CONTROL

    CHECK
    GO
    GOODBYE
```

Figure 21. Input for simple pipeline with surge tank.

Table 4. Element list for pipe and surge tank system.

Element I.D.	Type	Node Location	U.S. Node	D.S. Node
HW	Head Boundary	100	—	—
PIPE	Conduit	—	100	200
—	Simple Junction	200	—	—
DUM1	Dummy Conduit	—	200	300
TANK	Surge Tank	300	—	—
DUM2	Dummy Conduit	—	200	400
FBC	Flow Boundary	400	—	—

5.3 Oahe Pumping and Power Plant

The next example illustrates the data input for a very complex system, the proposed Oahe power and pumping plant (Resource Analysis 1978). The schematic diagram shown in Figure 22 is taken directly from this source, with some minor modifications. Gate valves upstream of the pumps are not included here because they are not operated as part of any of the simulations. Table 5 shows the element list. Program input is shown in Figure 23.

Some particular features of the input data worth noting are: (1) all six pumps, even those of different sizes, use the same dimensionless characteristics. (2) All six valves use the same characteristics. (3) The boundary element CANL appears at three separate node locations. Though it is not generally recommended to use the same element at different locations, in this case it is not confusing because the same physical component, the canal, is represented at each location. An alternative representation would show a junction at the end of the three discharge lines, with a dummy conduit leading from the junction to a single CANL element. This has the disadvantage of adding two new branches to the system. (4) With such a large, complex system, it is easy to forget the simple junctions when preparing the element list and the SYSTEM command. Care must be taken in this regard. (5) The specified operating conditions for this simulation are load rejection at the turbine and simultaneous shutdown of four pumps with closure of the discharge valves. The two larger pumps are not operated and their discharge valves are closed.

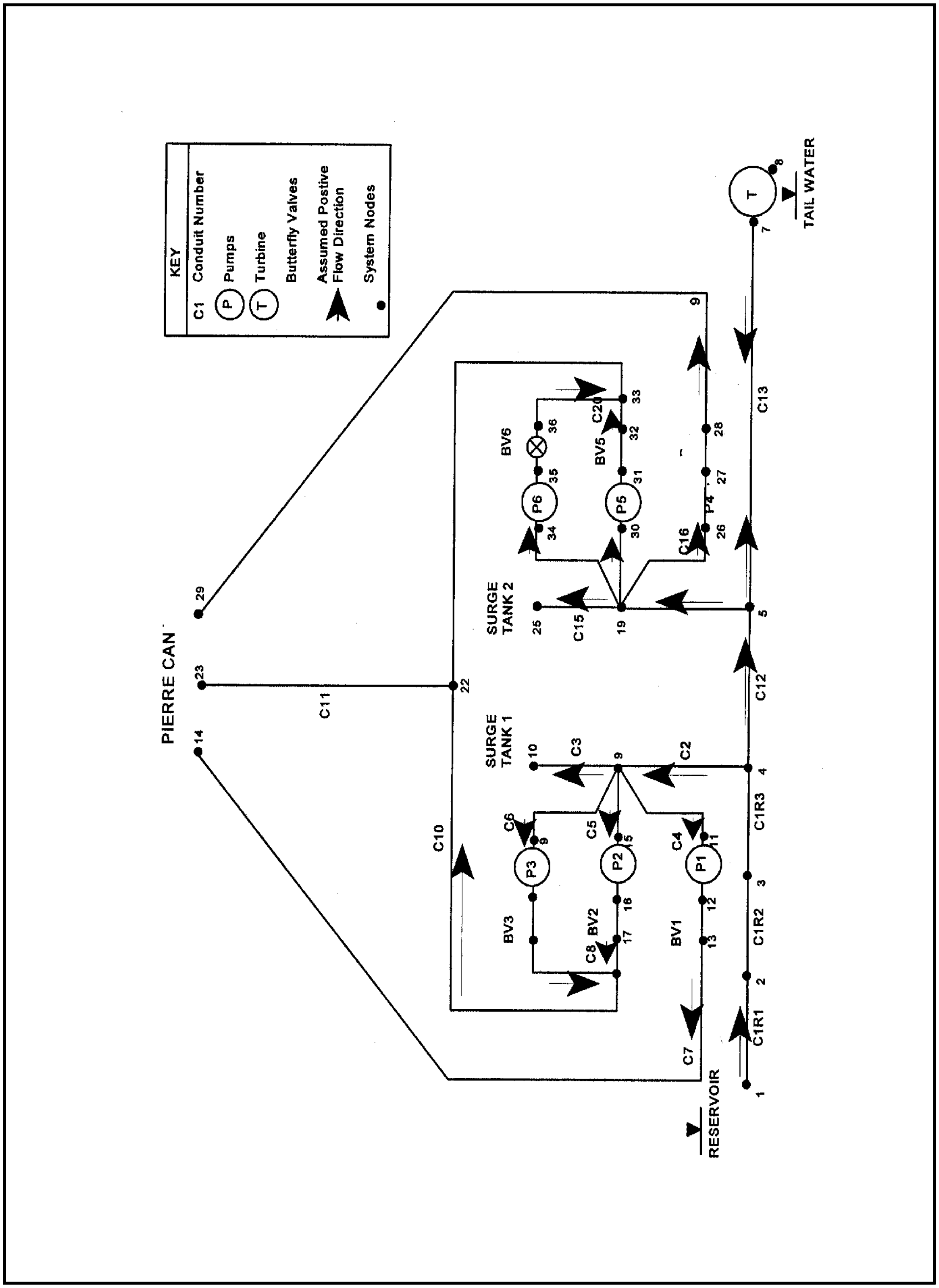


Figure 22. Proposed Oahe installation.

Table 5. Element list for Oahe power and pumping system.

Element I.D.	Type	Node Location	U.S. Node	D.S. Node
HW	Head Boundary	1	—	—
C1R1	Conduit	—	1	2
C1R2	Conduit	—	2	3
C1R3	Conduit	—	3	4
HLJ1	Tee Junction	4	—	—
C12	Conduit	—	4	5
HLJ2	Turbine	5	—	—
C13	Conduit	—	5	7
T	Turbine	—	7	8
TW	Head Boundary	8	—	—
C2	Conduit	—	4	9
—	Simple Junction	9	—	—
C3	Conduit	—	9	10
DC1	Diameter Change	10	—	—
ST1	Surge Tank	10	—	—
C4	Conduit	—	9	11
P1	Pump	—	11	12
BV1	Valve	—	12	13
C7	Conduit	—	13	14
CANL	Head Boundary	14	—	—
C5	Conduit	—	9	15
P2	Pump	—	15	16
BV2	Valve	—	16	17
C8	Conduit	—	17	18
—	Simple Junction	18	—	—
C6	Conduit	—	9	19
P3	Pump	—	19	20
BV3	Valve	—	20	21
C9	Conduit	—	21	18
C10	Conduit	—	18	22
—	Simple Junction	22	—	—
C11	Conduit	—	22	23
CANL	Head Boundary	23	—	—
C14	Conduit	—	5	24
—	Simple Junction	24	—	—
C15	Conduit	—	24	25
DC2	Diameter Change	25	—	—
ST2	Surge Tank	25	—	—
C16	Conduit	—	24	26
P4	Pump	—	26	27
BV4	Valve	—	27	28
C19	Conduit	—	28	29
CANL	Head Boundary	29	—	—
C17	Conduit	—	24	30
P5	Pump	—	30	31
BV5	Valve	—	31	32
C20	Conduit	—	32	33
—	Simple Junction	33	—	—
C18	Conduit	—	24	34
P6	Pump	—	34	35
BV6	Valve	—	35	36
C21	Conduit	—	36	33
C22	Conduit	—	33	22

```
OAHE POWER PLANT - LOAD REJECTION
C    SYSTEM CONNECTIVITY

SYSTEM
EL HW AT 1
EL C1R1 LINK 1  2
EL C1R2 LINK 2  3
EL C1R3 LINK 3  4
EL HLJI AT 4 RISER 9
EL C12 LINK 4  5
EL HLJ2 AT 5 RISER 24
EL C13 LINK 5 7
EL T LINK 7  8
EL TW AT 8
EL C2 LINK 4   9
JUNCTION AT 9
EL C3 LINK 9 10
EL DC1 AT 10
EL ST1 AT 10
EL C4 LINK 9 11
EL Pl LINK 11 12
EL BVI LINK 12 13
EL C7 LINK 13 14
EL CANL AT 14
EL C5 LINK 9 15
EL P2 LINK 15 16
EL BV2 LINK 16 17
EL C8 LINK 17 l8
JUNC AT 18
EL C6 LINK 9 19
EL P3 LINK 19 20
EL BV3 L INK 20 21
EL C9 LINK 21 18
EL C10 LINK 18 22
JUNC AT 22
EL C11 LINK 22 23
EL CANL AT 23
EL C14 LINK 5 24
JUNC AT 24
EL C15 LINK 24 25
EL OC2 AT 25
EL ST2 AT 25
EL C16 LINK 24 26
EL P4 LINK 26 27
EL BV4 LINK 27 28
EL C19 LINK 28 29
EL CANL AT 29
EL C17 LINK 24 30
EL P5 LINK 30 31
EL BV5 LINK 31 32
EL C20 LINK 32 33
JUNC AT 33
EL Cl8 LINK 24 34
EL P6 LINK 34 35
EL BV6 LINK 35 36
EL C21 LINK 36 33
EL C22 LINK 33 22
FINISH
```

Figure 23. Input for Oahe Power Plant.

```
C ELEMENT PROPERTIES

    RESERVOIR ID HW ELEV 1618.5 FINI
    CONDUIT ID C1R1 LENGTH 1387.50 DIAMETER 24. CELERITY 4730 FRICTION .008
      NUMSEG 3
      ENDLOSS AT HW CPLUS .525 CMINUS 1.24
      FINISH
    COND ID CIR2  LENG 1263.7 DIAM 24. CELE 3950 FRIC .008 NUMSEG 3 FINI
    COND 1D CIR3 LENG 1135.7 DIAM 26. CELE 2840 FRIC .008 NUMSEG3 FINI
    TJUNCTION ID HLJI    FILLET 0 FINI
    COND ID C12  LENG 39 DIAM 24. CELE 2501.6 FRIC .008 FINI
    TJUNCTION ID HLJ2   FILLET 0 FINI
    COND ID C13 LEN 108.1 DIAM 24.  CELE 2456.6 FRIC .008 FINI
    TURBINE ID T TYPE 1 SYNCSPD 100 WR2 148000000. FRICTION 250.
      WINDAGE 750 DIAMETER 20 FINISH
    RESE ID TW ELEV 1426.1 FINI
    COND ID C2 LENG 35 DIAM 17. CELE 2840 FRIC .008 FINI
    COND ID C3 LENG 46 DIAM 17. CELE 2840 FRIC .008 FINI
    DCHANGE ID DCI CPLUS 3.2 CMINUS 2.8 FINI
    SURGETANK ID ST1 ELBOTTOM 1515 ELTOP 1660 DIAM 70. CELE 2840 FRIC 0.02 FINI
    COND ID C4 LENG 208 DIAM 11.5 CELE 2434.2 FRIC .008 FINI
    PUMP ID Pl TYPE I RQ 1000 RHEAD I5O RSPEED 250 RTORQUE 286000
      WR2 3220000. FINISH
    VALVE ID BVI TYPE 1  DIAM 10. FINI
    COND ID C7 LENG 3210 NUMSEG 8 DIAM 8. CELE 2762.2 FRIC .009
      ENDLOSS AT CANL CPLUS 1.0 CMINUS 0~5
      FINI
    RESE ID CANL ELEV 1716 FINI
    COND ID C5 LENG 206 DIAM 7. CELE 2450.6 FRIC .010 FINI
    PUMP ID P2 TYPE I RQ 300 RHEAD 150 RSPEED 450  RTOROUE 47670
       WR2 250000 FINISH
    VALVE ID BV2 TYPE 1 DIAM 5.5 FINI
    COND ID C8 LENG 133.5 DIAM 5.5 CELE 2668.3 FRIC 0.012 FINI
    COND ID C6 LENG 204 DIAM 7. CELE 2450.6 FQ1C .010 FINI
    PUMP ID P3 AS P2 FINI
    VALVE ID BV3 TYPE 1 DIAM 5.5 FINI
    COND ID C9 AS C8 LENG 141.8 FINI
    COND ID C10 AS C7 LENG 77.3 FINI
    COND ID Cll LENG 3000 NUMSEG 7 DIAM 10.B3 CELE 2854.9 FRIC .008
      ENDLOSS AT CANL CPLUS 1.0 CMINUS 0.5
      FINI
    COND ID C14 AS C2 FINI
    COND ID C15 AS C3 FINI
   DCHANGE ID DC2 CPLUS 3.2 CMINUS 2.8 FINI
    SURGETANK ID ST2 AS ST1 FINI
    COND ID Cl6 AS C4 FINI
    PUMP ID P4 AS Pl FINI
    VALVE ID BV4 TYPE 1 DIAM 10. FINI
    COND ID C19 AS C7 FINI
    COND ID C17 AS C5 FINI
    PUMP ID P5 AS P2 FINI
    VALVE ID BVS TYPE 1 DIAM 5.5 FINI
    COND ID C20 AS C8 FINI
    COND ID Cl8 AS C6 FINI
    PUMP ID P6 AS P2 FINI
    VALVE ID BV6 TYPE 1 DIAM 5.5 FINI
   COND ID C21 AS C9 FINI
    COND ID C22 AS C10 FINI
```

Figure 23 continued.

```
C   VALVE, PUMP, AND TURBINE CHARACTERISTICS

V CHARACTERISTICS TYPE 1
   ANGLE 0.0 DISCOEF 1.71 A 5.0 DC 1.63 A 10.0 DC 1.40 A 15.0 DC 1.30
   A 20.0 DC 1.025 A 25.0 DC 0.775 A 30.0 DC 0.61 A 35.0 DC 0.485
   A 40.0 DC 0.375 A 45.0 DC 0.280 A 50.0 DC 0.205 A 55.0 DC 0.165
   A 60.0 DC 0.125 A 65.0 DC 0.090 A 70.0 DC 0.055 A 75.0 DC 0.030
   A 80.0 DC 0.017 A 85.0 DC 0.012 A 88.0 DC 0.005 A 90.0 DC 0.00
    FINISH

P CHARACTERISTICS TYPE 1
   SRATIO
         -1.5    -1.25   -1.0    -0.75   -0.5    -0.25   0.0     0.25
         0.50    0.75    1.00
   QRATIO
           -1.1  -0.9    -0.7    -0.5    -0.25   0.0     0.25    0.50
           .75   1.00    1.25    1.50
   HRATIO
         0.95    0.90    0.93    1.00    1.25    1.40    1.55    1.75
         2.10    2.76    3.60    0.79    0.64    0.60    0.65    0.73
         0.88    1.07    1.30    1.60    2.20    2.80    0.76    0.54
         0.40    0.36    0.40    0.50    0.65    0.85    1.20    1.75
         2.35    0.74    0.52    0.35    0.23    0.20    0.25    0.34
         0.50    0.64    1.34    2.00    0.74    0.52    0.34    0.18
         0.12    0.12    0.15    0.27    0.53    1.10    1.70    0.74
         0.52    0.33    0.18    0.06    0.02    0.0     0.13    0.40
         0.94    1.55    0.65    0.41    0.22    0.04    0.00    -0.10
         -0.12   0.06    0.35    0.83    1.48    0.44    0.20    0.00
         -0.20   -0.32   -0.40   -0.33   -0.08   0.25    0.74    1.37
         0.00    -0.30   -0.50   -0.70   -0.85   -0.87   -0.70   -0.40
         0.05    0.60    1.25    -0.65   -1.00   -1.30   -1.40   -1.45
         -1.38   -1.20   -0.80   -0.30   0.30    1.00    -1.50   -1.85
         -2.10   -2.30   -2.30   -2.00   -1.70   -1.20   -0.74   -0.11
         0.64    -2.55   -2.85   -2.90   -3.40   -3.40   -2.73   -2.20
         -1.6    -1.3    -0.6    0.25
   TRATIO
         0.10    0.40    0.80    1.10    1.40    1.65    1.90    2.08
         2.25    2.48    3.00    -0.32   0.0     0.33    0.63    0.82
         1.10    1.30    1.50    1.70    2.00    2.45    -0.70   -0.31
         0.00    0.20    0.40    0.60    0.80    0.92    1.20    1.60
         2.10    -1.00   -0.60   -0.29   -0.04   0.13    0.27    0.40
         0.53    0.76    1.20    1.70    -1.38   -0.92   -0.56   -0.25
         -0.05   0.05    0.13    0.20    0.43    0.78    1.25    -1.80
         -1.23   -0.80   -0.46   -0.25   -0.11   0.00    0.12    0.26
         0.57    1.00    -2.70   -2.0    -1.3    -0.90   -0.60   -0.30
         -0.12   0.07    0.25    0.53    0.90    -3.50   -2.75   -2.10
         -1.60   -1.20   -0.77   -0.34   -0.04   0.25    0.58    0.97
         -4.40   -3.60   -3.00   -2.35   -1.90   -1.35   -0.80   -0.32
         0.15    0.55    1.02    -5.40   -4.60   -4.00   -3.35   -2.70
         -2.00   -1.40   -0.80   -0.10   0.44    1.00    -6.10   -5.50
         -4.90   -4.30   -3.50   -2.75   -2.15   -1.30   -0.54   0.19
         0.85    -6.50   -6.30   -5.70   -5.20   -4.30   -3.60   -2.90
         -2.00   -1.20   -0.20   0.50
 FINISH
```

Figure 23 (continued)

```
TCHARACTERISTICS TYPE 1
  GATEPOS
          0.00    10.00   25.00   43.38   53.23   65.08   70.50   75.92
          81.34   86.77   97.61
  HEAD
          0.0     40.0    80.0    100.0   120.0   130.0   140.0   150.0
          160.0   170.0   180.0   190.0   200.0   210.0   220.0
   Q
          0.0     54.0    165.0   1086.0  1828.0  2390.0  2844.0  3090.0
          3300.0  3562.0  3968.0  0.0     267.0   637.0   1718.0  2514.0
          3195.0  3622.0  3920.0  4196.0  4481.0  4934.0  0.0     480.0
          1110.0  2350.0  3200.0  4000.0  4400.0  4750.0  5093.0  5400.0
          5900.0  0.0     585.0   1345.0  2666.0  3543.0  4402.0  4789.0
          5165.0  5540.0  5859.0  4383.0  0.0     693.0   1581.0  2982.0
          3886.0  4805.0  5178.0  5580.0  5990.0  6319.0  6866.0  0.0
          779.0   1749.0  3158.0  4079.0  5035.0  5440.0  5853.0  6261.0
          6584.0  7151.0  0.0     889.0   2000.0  3327.0  4259.0  5258.0
          5685.0  6104.0  6506.0  6835.0  7419.0  0.0     996.0   2275.0
          3494.0  4434.0  5478.0  5922.0  6337.0  6724.0  7073.0  7673.0
          0.0     1015.0  2311.0  3650.0  4615.0  5693.0  6140.0  6561.0
          6951.0  7306.0  7926.0  0.0     1046.0  2382.0  3782.0  4792.0
          5903.0  6324.0  6765.0  7259.0  7535.0  8168.0  0.0     1076.0
          2451.0  3906.0  4948.0  6094.0  6501.0  6959.0  7417.0  7761.0
          8409.0  0.0     1106.0  2519.0  4032.0  5098.0  6259.0  6672.0
          7145.0  7591.0  7982.0  8643.0  0.0     1143.0  2584.0  4160.0
          5242.0  6405.0  6836.0  7326.0  7795.0  8194.0  8870.0  0.0
          1162.0  2648.0  4284.0  5377.0  6554.0  6995.0  7507.0  7985.0
          8397.0  9095.0  0.0     1190.0  2710.0  4403.0  5505.0  6702.0
          7148.0  7687.0  8167.0  8590.0  9318.0
    EFFICIENCY
          0.000   0.380   0.560   0.563   0.567   0.570   0.573   0.657
          0.707   0.770   0.835   0.000   0.380   0.560   0.570   0.594
          0.643   0.677   0.739   0.779   0.823   0.858   0.000   0.380
          0.560   0.605   0.727   0.758   0.781   0.821   0.851   0.876
          0.881   0.000   0.380   0.660   0.680   0.793   0.815   0.833
          0.862   0.887   0.903   0.893   0.000   0.380   0.560   0.755
          0.860   0.873   0.885   0.903   0.923   0.929   0.904   0.000
          0.380   0.560   0.793   0.879   0.892   0.904   0.922   0.933
          0.931   0.907   0.000   0.380   0.560   0.825   0.894   0.907
          0.918   0.934   0.937   0.928   0.906   0.000   0.380   0.560
          0.849   0.904   0.916   0.927   0.938   0.936   0.926   0.904
          0.000   0.380   0.560   0.864   0.908   0.922   0.934   0.936
          0.931   0.924   0.901   0.000   0.380   0.560   0.867   0.906
          0.925   0.936   0.931   0.915   0.920   0.897   0.000   0.380
          0.560   0.863   0.905   0.925   0.930   0.925   0.916   0.915
          0.892   0.000   0.380   0.560   0.859   0.902   0.920   0.922
          0.918   0.914   0.909   0.887   0.000   0.380   0.560   0.855
          0.900   0.911   0.915   0.911   0.905   0.902   0.881   0.000
          0.380   0.560   0.852   0.888   0.903   0.907   0.903   0.898
          0.896   0.875   0.000   0.380   0.560   0.848   0.896   0.894
          0.901   0.895   0.890   0.890   0.870
    FINISH
```

Figure 23 (continued)

```
C           OPERATION SPECIFICATIONS

OPTURB ID T REJECT VSCHEDULE 10 FINI

OPPUMP
            ID P1 OFF
            ID P2 SHUTOFF
            ID P3 SHUTOFF
            ID P4 OFF
            ID P5 SHUTOFF
            ID P6 SHUT
            FINISH

VALVE ID BV1 VSCHEDULE 1 FINI
VALVE ID BV2 VSCHED 3 FINI
VALVE ID BV3 VSCHED 3 FINI
VALVE ID BV4 VSCHED 1 FINI
VALVE ID BV5 VSCHED 3 FINI

SCHEDULE
            VSCHEDULE 1 TIME 0 ANGLE 90
            VSCH 2 T 0 A 0
            VSCH 3
            T 0.0 A 0.0 T 10 A 17 T 20 A 33 T 30 A 50 T 40 A 67 T 50 A 83 T 55 A 88
            T 58 A 89 T 59 A 89.6 T 60 A 90
            VSCH4
             T 0 A 90 T 10 A 73 T 20 A 57 T 30 A 40 T 40 A 23 T 50 A 7 T 55 A 2 T 58 A1
            T 59 A 0.4 T 60 A 0
            FINISH

SCHEDULE
            VSCHED 10 T 0 G 77 T 0.3 G 75,85 T 0.5 G 71.84 T 2.5 G 24.72 T 3.1 G 11.47
            T 3.6 G 5.75 T 4.1 G 2.87 T 4.6 G 1.15 T 5.1 G 0.0
            VSCHED 11 T 0 G 97.6 T 0.6 G 96.1 T 1.0 G 91.1 T 5.0 G 31.3 T 6.2 G 14.5
            T 7.2 G 7.29 T 8.2 G 3.63 T 9.2 G 1.45 T 10.2 G 0
            VSCHED 12 T 0 G 97.6 T 0.5 G 97.6 T 0.8 G 97.6 T 2.0 G 83.0 T 4.0 G 59.9
            T 6.0 G 35.0 T 7.0 G 25.0 T 7.5 G 20.0 T 8.0 G 15.0 T 8.6 G 10.0 T 9.0 G 6.0
            T 9.5 G 3.0 T 10.0 G 1.0 T 11.5 G 0.0
            FINISH

C           OUTPUT REQUESTS

HISTORY
            NODE 7 Q HEAD PIEZ
            ELEM T SPEED
            NODE 4 TONODE 3 Q
            FINISH
PLOT
            NODE 7 Q HEAD PIEZ
            ELEM T SPEED POSITION
            ELEM ST1 ELEV
            FINI
DISPLAY OFF FINI

C           COMPUTATIONAL PARAMTERS

CONTROL
            DTCOMP 0.1 DTOUT 1.0 TMAX 12
            DTCOMP 1.0 DTOUT 5.0 TMAX 60.
            DTCOMP 5.0 DTOUT 10.0 DTMAX 400.
            FINI
C           EXECUTION CONTROL

CHECK
GO
GOODBYE
```

Figure 23 (continued)

5.4 Clinton Lake Pumping Plant—Case Study

The following case study was provided by the Omaha District of the Corps of Engineers.

5.4.1 Introduction and Purpose

The Clinton Lake pumping plant is owned and operated by the City of Lawrence (Kansas). The plant's water supply is obtained from the normal discharge line of the Corps of Engineers Clinton Lake Reservoir on the Wakarusa River. Personnel from the Kansas City District and Missouri River Division noticed vibration at a manhole on the normal discharge line and were concerned that it was a result of water hammer caused by operation of the pumping plant. The Omaha District, because of their experience with hydraulic transients, was recommended to perform a hydraulic transient study for the plant.

The scope of this study was limited to the determination of hydraulic pressures in the system resulting from transient operation of the pumping plant and reservoir discharge gate.

5.4.2 Study Method

The Corps of Engineer's WHAMO computer program was used to model the pumping plant and its operation including intake and discharge lines and the reservoir's normal discharge valve. The most recent version of the WHAMO program can model all of the elements of the Clinton pumping plant configuration, except for the air valves on the pump discharge line. However, as of this time, the plotting routine is not available for use and printed output data were manually plotted.

Omaha District personnel made a field trip to gather data on plant operation and verify drawings supplied by the Kansas City District. It was also necessary to get name tag data off the plant equipment and get dimensional data off the City of Lawrence drawings for the plant's discharge line.

Transient pressures for a pumping plan system are dependent on the particular characteristics of the pump. The 4-quadrant performance curves necessary for best model simulation of the pumps used at the plant were not available. Available characteristics for constant speed operation of those units were modified using hydraulic machinery equations for first quadrant representation. It was necessary to model the pumps in the reverse direction of rotation so 4-quadrant curves for pumps of both higher and lower specific speeds were substituted. This was done assuming that the actual performance of the Clinton Lake pumping plant pumps would fall between those two. The Clinton Lake pumps have a specific speed of 3111. The speeds of the other two are 3852 and 1935.

It was assumed that the most severe water hammer pressures would result from intentional pumping shutdowns or accidental tripout of the pumps. Cases to be modeled would include variations of those cases with the pump's pneumatic shutoff valves and/or the check valve failing. Cases were modeled with the reservoir's normal discharge valve both open and closed. Transient operation of the normal discharge valve was not considered since it is manually operated and cannot be moved quickly.

5.4.3 Plant Configuration and Operation

The pumping plant intake line taps on to the normal discharge line of the reservoir near the downstream end (see Figure 25). The plant has three pumps each with a rated capacity of 7.80 cfs and provisions for one more pump. Each has an air operated shutoff valve that closes in approximately 27 seconds during either normal or emergency shut-downs. During pump trip-offs, the pumps are still rotating in the pumping direction after the valves have closed. Inertia of the pumps and their motors was obtained from manufacturers.

The plants discharge line is 36 inches in diameter, 9200 feet long and has a one-way check valve with an 8 in. bypass near the pumps. The discharge line has high points at distances of approximately 900 feet and 2500 feet from the plant. Both high points are equipped with air valves. The valves are designed to let air in, in case of a water column separation, and air out for positive pressure conditions. The reservoir normal discharge valve is located at the end of the reservoir's discharge line. It is a manually operated gate type valve. Maximum discharge is limited to 35 cfs. Reservoir discharge and pumping usually occur simultaneously.

5.4.4 Results And Recommendations

Results of this study indicate that no abnormal pressure surges can occur in the reservoir discharge line or the pumping plant intake line. For the cases modeled, a pressure of 27 feet above reservoir elevation is the maximum pressure expected and that would occur in the pump intake line near the pumps, during a power loss with all three units previously pumping and with the pump shut off valves working. Failure of the valves would not increase that pressure.

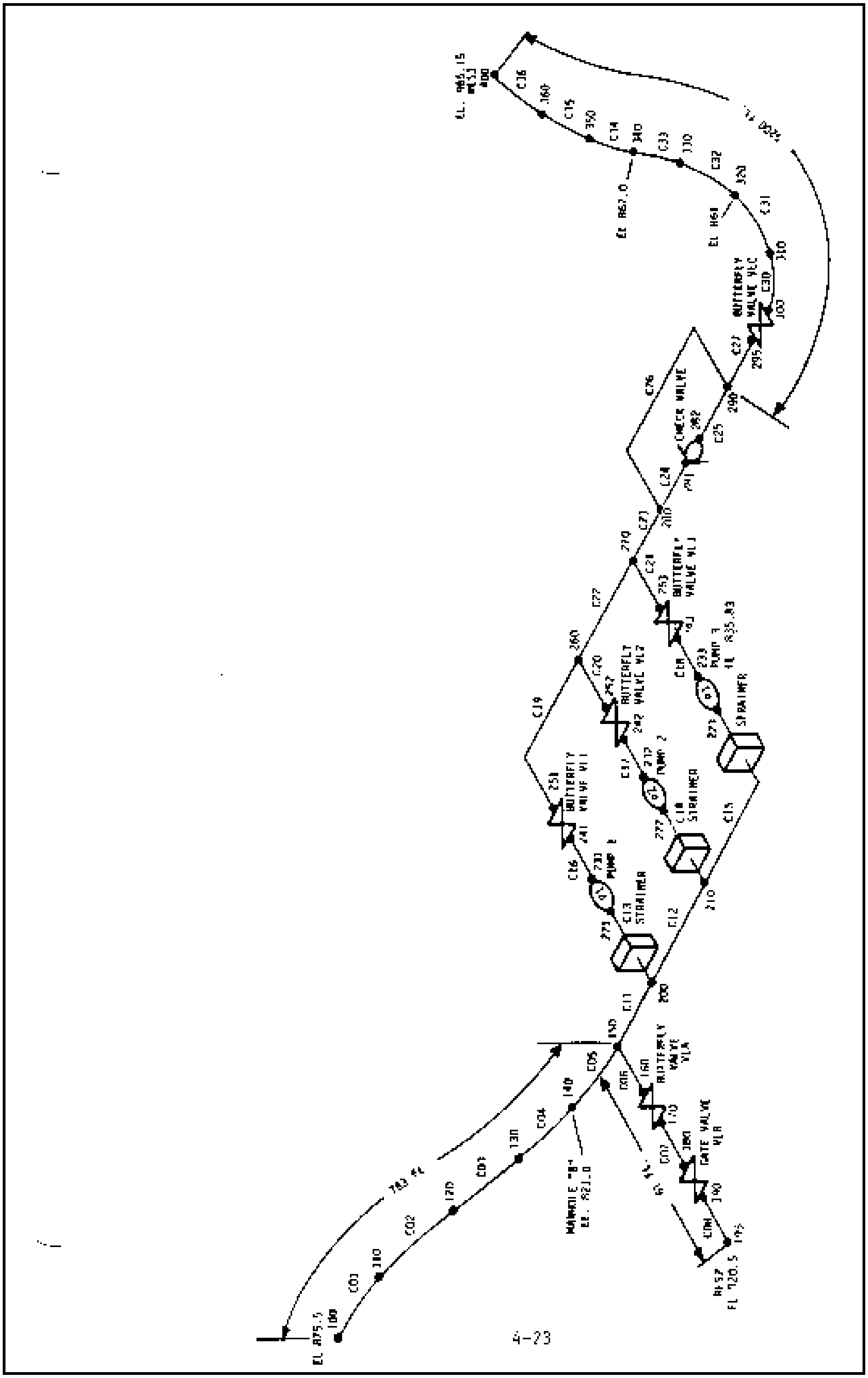


Figure 24. Schematic of Clinton Lake Pumping Plant.

Results also indicate that no abnormal pressure surges would occur in the pump discharge line. The lowest pressure expected would be 10 feet below atmospheric pressure and would occur at the first air valve after a power loss of all three pumps. The highest pressure predicted by the WHAMO program would be 65 feet above static head and would occur near the pumps. Actual maximum pressure could be different if the air valves let air into the system because the WHAMO program does not model mixed flow conditions. It is not likely that much air would enter the line because the time that the line is below atmospheric pressure is short.

The results indicate the following conclusions as far as valve operation is concerned. (1) If the reservoir discharge valve is open, pressures throughout the system are less severe. (2) The pneumatic shutoff valves for the pumps are only to prevent wasting water, pressures are actually slightly higher with them closing as designed. (3) The system was modeled with the check valve slamming shut after a power loss and that was not a critical case.

The system was modeled for the most critical case which is a pumping power loss with all valves working properly and with the 4th pump installed. Maximum and minimum pressures become only slightly more severe.

5.4.5 Confidence Level of Study

The most significant factors which could cause errors in the study results would be inaccuracy in modeling the pump characteristics and not accurately modeling closure of the pump shutoff valves.

As stated previously, three different sets of pump characteristics were used in the WHAMO program. It is unlikely that the actual pump characteristics would produce water hammer characteristics more severe than obtained for the most severe of those three. The results are very similar for all three.

Exact valve closure rates were not measured in the field but a rate of as quick as full closure in 14 seconds could be made with little increase in pressures. That rapid a closing rate is probably not possible. Actual valve closure rate is about 27 seconds.

Severe water hammer in the pumping plant discharge line is not likely and could not occur in the pumping plant intake line as a result of operation of the pumps.

5.4.6 Cases Modeled

The following is a description of the cases modeled.

Normal Pump Shutoffs: Case 1. This case assumes that only one pump is being used and is then shut off. The reservoir discharge valve is open. Patterson pump characteristics are used. Not a critical case.

Case 2. Same as Case No. 1 except that the reservoir discharge valve is closed. No critical pressures but most severe for a normal shut off.

Case 3. This case assumed that all three pumps are running and one of them is shut off. The reservoir discharge valve is open. Not a critical case.

Case 4. Same as Case No. 3 except that the reservoir discharge valve is closed.

Pump Trip-Offs (All Valves Work): Case 5. This case assumes all three pumps lose power simultaneously and pump shutoff valves close in 27 seconds. The discharge valve is open and Patterson pump characteristics are used. Not a critical case.

Case 6. Same as Case No. 5 except that the reservoir discharge valve is closed. Produces most severe pressure in the pump discharge line.

Case 7. Same as Case No. 5 but uses Oahe pump characteristics; not a critical case.

Case 8. Same as Case No. 5 except uses Oahe pump characteristics and the reservoir discharge valve is closed. Produces the most severe pressures in the pump intake line. Some of the results are plotted in Figures 25 and 26. WHAMO input data are shown for this case in Figure 27.

Case 9. Same as Case No. 5 except uses Kittredge pump characteristics. Not a critical case.

Case 10. Same as Case No. 5 except uses Kittredge pump characteristics and the reservoir discharge valve is closed.

Case 11. This case is the same as Case No. 6 but assumes that the 4th pump is installed and initially pumping.

Case 12. This case is the same as Case No. 8 but assumes that the 4th pump is installed and initially pumping.

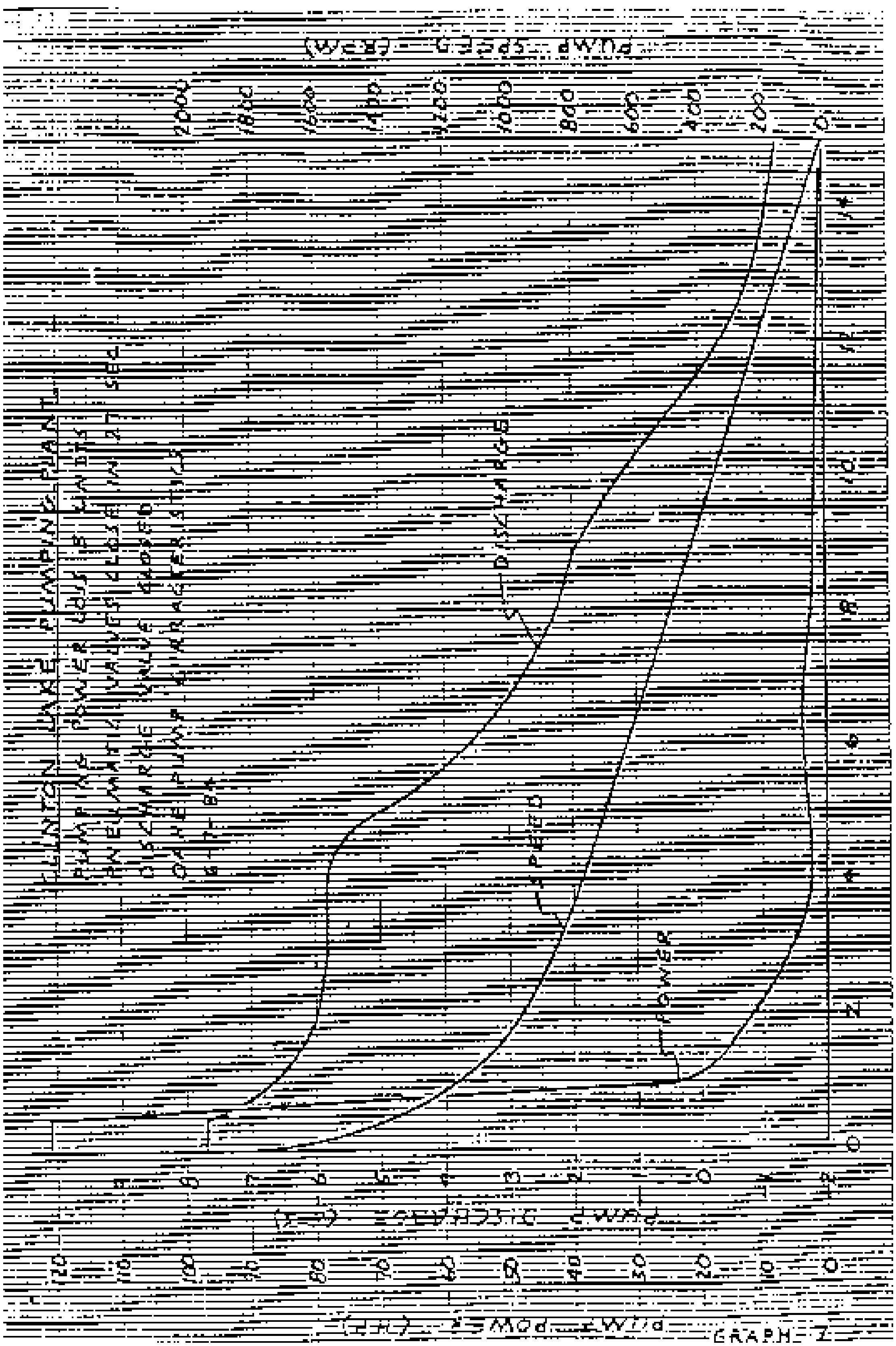


Figure 25. Simulated pump behavior (Case 8).

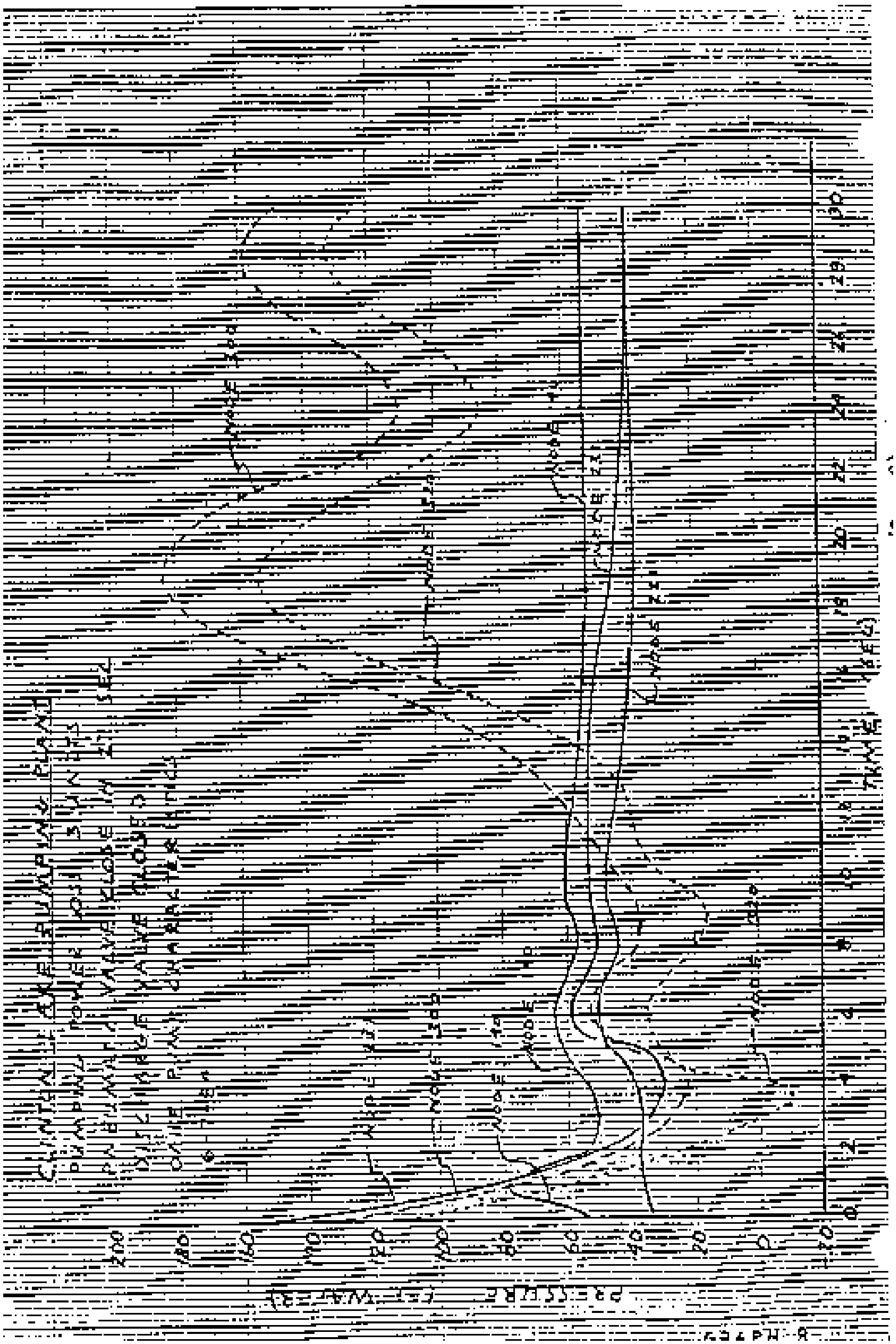


Figure 26. Simulated pipeline pressures (Case 8).

```
CLINTON LAKE   POWER LOSS 3 PUMPS VALVES OPER, OAHE CHAR., VLB CLOSED

SYSTEM
          EL RES1 AT 100
          EL RES2 AT 195
          EL RES3 AT 400
          EL C01 LINK 100 110
          EL C02 LINK 110 120
          EL C03 LINK 120 130
          EL C04 LINK 130 140
          EL C05 LINK 140 150
          EL C06 LINK 150 160
          EL VLA LINK 160 170
          EL C07 LINK 170 180
          EL VLB LINK 180 190
          EL C08 LINK 190 195
          EL C11 LINK 150 200
          EL C12 LINK 200 210
          EL C13 LINK 200 221
          EL C14 LINK 210 222
          EL C15 LINK 210 223
          EL P1 LINK 221 231
          EL P2 LINK 223 233
          EL C16 LINK 231 241
          EL C17 LINK 232 242
          EL C18 LINK 233 243
          EL VL1 LINK 241 251
          EL VL2 LINK 242 252
          EL VL3 LINK 243 253
          EL C19 LINK 251 260
          EL C20 LINK 252 260
          EL C21 LINK 253 270
          EL C22 LINK 260 270
          EL C23 LINK 270 280
          EL C24 LINK 280 281
          EL VCK LINK 281 282
          EL C25 LINK 282 290
          EL C26 LINK 280 290
          EL C27 LINK 290 295
          EL VLC LINK 295 300
          EL C30 LINK 300 310
          EL C31 LINK 310 320
          EL C32 LINK 320 330
          EL C33 LINK 330 340
          EL C34 LINK 340 350
          EL C35 LINK 350 360
          EL C36 LINK 360 400
          JUNC AT 150
          JUNC AT 280
          JUNC AT 290
          JUNC AT 200
          JUNC AT 210
          JUNC AT 260
          JUNC AT 270
          NODE 320 ELEV 861
          NODE 340 ELEV 867
 FINISH
```

Figure 27. WHAMO input for Clinton Lake.

```
C   ELEMENT PROPERTIES
          RESERVOIR ID RES1 ELEV 875.5 FINISH
          RESERVOIR ID RES2 ELEV 828.5 FINISH
          RESERVOIR ID RES3 ELEV 965.75 FINISH

          COND ID C01 LENG 79.3 D1AM 3.5 CELE 4000 FRIC 0.0065
                    ADDEDLOSS AT 000.0 CPLUS 0.50 CMINUS 1.00
                    ADDEDLOSS AT 14.8 CPLUS 0.42  CMINUS 0.42
                    ADDEDLOSS AT 53.3 CPLUS 0.42  CMINUS 0.42      FINISH
          COND ID C02 LENG 33.0 DIAM 3.5 CELE 4000 FRIC 0.0065
                    ADDEDLOSS AT 000.0 CPLUS 0.60 CMINUS 0.60
                    ADDEDLOSS AT 9.0 CPLUS 0.24  CMINUS 0.24
                    ADDEDLOSS AT 18.0 CPLUS 0.24 CMINUS 0.24      FINISH
          COND ID C03 LENG 10.0 CELE 4000 FRIC 0.0065
                    VARI    DIST   0.0    AREA 9.62
                    VARI    DIST 10.0    AREA 7.06
                    ADDEDLOSS AT 000.0 CPLUS 0.20 CMINUS 0.20     FINISH
          COND ID C04 LENG 526.8 D1AM 3.0 CELE 4000 FRIC 0.0065
                    ADDEDLOSS AT 526.8 CPLUS 0.42 CMINUS 0.42   FINISH
          COND ID C05 LENG 113.8 D1AM 3.0 CELE 4000 FRIC 0.0065
                    ADDEDLOSS AT 8.0 CPLUS 0.20 CMINUS 0.20
                    ADDEDLOSS AT 84.4 CPLUS 0.20  CMINUS 0.20
                    ADDEDLOSS AT 91.7 CPLUS 0.42  CMINUS 0.42      FINISH
          COND ID C06 LENG 2.0 D1AM 2.0 CELE 4000 FRIC 0.0065     FINISH
          COND ID C07 LENG 54.6 D1AM 2.0 CELE 4000 FRIC 0.0065
                    ADDEDLOSS AT 9.2 CPLUS 0.15 CMINUS 0.15
                    ADDEDLOSS AT 14.2 CPLUS 0.20  CMINUS 0.20
                    ADDEDLOSS AT 32.4 CPLUS 0.20  CMINUS 0.20      FINISH
          COND ID C08 LENG 4.0 D1AM 2.0 CELE 4000 FRIC 0.0065
                    ADDEDLOSS AT 4.0 CPLUS 1.0 CMINUS 1.0   FINISH
          COND ID C11 LENG 10.0 D1AM 3.0 CELE 4000 FRIC 0.0065      FINISH
          COND ID C12 AS C11 FINISH
          COND ID C13 LENG 10.0 D1AM 1.0 CELE 4000 FRIC 0.0065
                    ADDEDLOSS CPLUS 3.00 CMINUS 3.00     FINISH
          COND ID C14 AS C13 FINISH
          COND ID C15 LENG 20.0 D1AM 1.0 CELE 4000 FRIC 0.0065
                    ADDEDLOSS CPLUS 3.75 CMINUS 3.75     FINISH
          COND ID C16 LENG 8.0 D1AM 0.8 CELE 4000 FRIC 0.0065   FINISH
          COND ID C17 AS C16 FINISH
          COND ID C18 AS C16 FINISH
          COND ID C19 LENG12.0 D1AM 0.8 CELE 4000 FRIC 0.0065
                    ADDEDLOSS CPLUS 2.50 CMINUS 2.50 FINISH
          COND ID C20 LENG 2.0 D1AM 0.8 CELE 4000 FRIC 0.0065
                    ADDEDLOSS CPLUS 1.80 CMINUS 1.80 FINISH
          COND ID C21 AS C20 FINISH
          COND ID C22 AS C11 FINISH
          COND ID C23 AS C11 LENG 6.0   FINISH
          COND ID C24 AS C11 LENG 2.0  FINISH
          COND ID C25 AS C24 FINISH
          COND ID C26 LENG 6.0 D1AM 0.7 CELE 4000 FRIC 0.0065
                    ADDEDLOSS CPLUS 1.50 CMINUS 1.50 FINISH
          COND ID C27 AS C11 LENG 4.0   FINISH
          COND ID C30 AS C11 LENG 277.0  FINISH
          COND ID C31 LENG 647.0 DIAM 3.0 CELE 4000 FRIC 0.0065 FINISH
          COND ID C32 AS C31 LENG 935.0  FINISH
          COND ID C33 AS C31 LENG 665.0  FINISH
          COND ID C34AS C31 LENG 1195.0  FINISH
```

Figure 27 (continued)

```
COND ID C35 AS C31 LENG 4895.0    FINISH
COND ID C36 AS C31 LENG 610.0    FINISH
VALVE ID VL1 BUTTERFLY DIAM 0.70 VSCHEDULE 1    FINISH
VALVE ID VL2 BUTTERFLY DIAM 0.70 VSCHEDULE 1    FINISH
VALVE ID VL3 BUTTERFLY DIAM 0.70 VSCHEDULE 1     FINISH
VALVE ID VLA BUTTERFLY DIAM 2.00 VSCHEDULE 4     FINISH
VALVE ID VLB GATE DIAM 2.00 VSCHEDULE 5   FINISH
VALVE ID VLC BUITERFLY DIAM 3.00 VSCHEDULE  4 FINISH
ONEWAY ID VCK DIAM 3.0 FINISH
PUMP ID P1 TYPE 1 RHEAD 110.0 RQ 7.80  RSPEED 1786.0
          RTORQUE 356.1     WR2 115.0          FINISH
PUMP ID P2 AS P1     FINISH
PUMP ID P3 AS P1  FINISH

C   PUMP CHARACTERISTICS

   PCHARACTERISTICS TYPE 1

   SRATIO -1.50 -1.25 -1.00 -0.75 -0.50 -0.25 0.00 0.25 0.50 0.75 1.00
   QRATIO -1.1 -0.9  -0.7   -0.5   -0.25 0.0   0.25  0.5   0.75  1.0     1.25
          1.50

 HRATIO          0.95  0.90  0.93  1.00  1.25  1.40  1.55  1.75  2.10  2.76   3.60
                 0.79  0.64  0.60   0.65  0.73  0.88  1.07  1.30  1.60  2.20   2.80
                 0.76  0.54  0.40   0.36  0.40  0.50  0.65 0.85   1.20  1.75   2.35
                 0.74  0.52  0.35   0.23  0.20  0.25  0.34  0.50  0.64  1.38   2.00
                 0.74  0.52  0.34  O.18  0.12  O.12  O.15  0.27  0.53  1.10  1.70
                 0.74  0.52  O.33 0.18   0.06  0.02  0.00  0.13   0.4O 0.94   1.55
                 0.65  0.41  0.22  0.08   0.00 -0.10 -0.12   0.06 0.35 0.83    1.480
                 0.44  0.20 0.00 -0.20   -0.32 -0.40 -0.33 -0.08 0.25 0.74 1.37
                 0.00  -0.30 -0.50 -0.70 -0.85 -0.87 -0.70 -0.40 0.05 0.60 1.25
                 -0.65 -1.00 -l.3O -1.40 -1.45 -l.38  -1.20 -0.80 -0.30 0.30 1.00
                 -1.50 -l.85  -2.10 -2.3O -2.30 -2.00 -1.70 -1.20 -0.74 -0.11 0.64
                 -2.55 -2.85 -2.90 -3.40 -3.40 -2.73 -2.20 -1.60  -1.30 -0.60  0.25
 TRATIO          0.10    0.40   0.80 1.10 1.40 1.65 l.9O 2.08 2.25 2.48 3.00
                 -0.32 0.00 0.33 0.63 0.82 1.10  1.30  1.50 1.70  2.00 2.45
                 -0.70 -0.31 0.00   0.20  0.40   0.60 0.80     0.92   1.20 1.60 2.10
                 -1.00   -0.60   -0.29   -0.04   0.13 0.27 0.40 0.53 0.78 1.20 1.70
                 -1.80  -1.23  -0.80  -0.46  -0.25  -0.11 0.00 0.12 0.26 0.57 1.00
                 -2.70 -2.00 -1.30 -0.90 -0.60 -0.30 -0.12 0.07 0.25 O.53 0.90
                 -3.50 -2.75 -2.10 -1.60 -1.20 -0.77 -0.34 -0.04 0.25  0.58 0.97
                 -4.40 -3.60 -3.00 -2.35 -1.90 -1.35 -0.80 -0.32 0.15  0.55 1.02
                 -5.40 -4.60 -4.00 -3.35 -2.70 -2.00  -1.40 -0.80  -0.10  0.44 1.00
                          -6.10 -5.50 -4.90 -4.30 -3.50 -2.75 -2.15 -1.30 -0.54   0.19 0.85
                 -6.50 -6.30 -5.70 -5.20 -4.30 -3.60 -2.90 -2.00 -1.20 -0.20 0.50

FINISH

C         PUMP OPERATION

          OPPUMP ID P1 SHUTOFF TOFF 0.0 FINISH
          OPPUMP ID P2 SHUTOFF TOFF 0.0 FINISH
          OPPUMP ID P3 SHUTOFF TOFF 0.0 FINISH
```

Figure 27 (continued)

```
C    VALVE OPERATION SCHEDULES.
            SCHEDULE
             VSCHEDULE 1   T 000 A 0.0
                                 T 10.0 A 60.0
                                 T 27.0 A 90.0
              VSCHEDULE 2 T 0.0 G 90.0
              VSCHEDULE 4 T 0.0 A 0.0
              VSCHEDULE 5 T 0.0 G 0.0
   FINISH

  HISTORY
   ELEM P1 SPEED POWER Q
   ELEM P2 SFEED POWER Q
   ELEM P3 SPEED POWER Q
   NODE 221 HEAD
   NODE 222 HEAD
   NODE 223 HEAD
   NODE 231 HEAD
   NODE 232 HEAD
   NODE 233 HEAD
   NODE 300 HEAD Q
   NODE 140 HEAD
   NODE 320 HEAD PRES
   NODE 340 HEAD PRES
   ELEM VLB POSITION
   ELEM VL1 POSITION
   ELEM VL2 POSITION
   ELEM VL3 POSITION
   NODE 190 Q
FINISH

CONTROL
   DTCOMP 0.5 DTOUT 0.5 TMAX 10.0
   DTCOMP 1.0 DTOUT 1.0 TMAX 50.0
   FINISH

 DISPLAY OFF FINISH
 DISPLAY ALL FINI

 CHECK
 GO
GOODBYE
```

Figure 27 (continued)

***Pump Trip-Offs (Pneumatic Valves Fail): Case 13*.** This assumes that all three pumps lose power and their shut-off valves fail to close. The reservoir discharge valve is open and uses Oahe pump characteristics. Not a critical case.

***Case 14*.** Same as Case No. 13 except that the reservoir discharge valve is closed.

***Case 15*.** Same as Case No. 13 except that the reservoir discharge valve is closed and uses Kittredge pump characteristics.

***Pump Trip-Off (Pneumatic Valves Fail and Check Valve Fails): Case 16*.** This assumes that all three pumps lose power and their shut-off valves fail to close. The check valve initially fails to close, then slams shut after 50 seconds. The reservoir discharge valve is closed.

5.5 Ponape Power Plant—Case Study

The following case study was provided by the North Pacific Division of the Corps of Engineers.

In 1983 the Corps of Engineers designed the Ponape Hydro-Electric Project located on the island of Ponape in Micronesia. The project consisted of building a 4600 ft. long penstock from an existing dam to the new powerhouse. The powerhouse has two small horizontal shaft Francis turbines directly coupled to synchronous generators. The two units are capable of producing a total of 1600 kW when operating under a net head of 200 feet.

Because of its length, the cost of the penstock was a significant fraction of the total project cost. To minimize the penstock cost, the turbine-generator equipment and control logic were selected to limit the maximum water hammer pressure to 130 percent of the maximum static pressure. No surge tank was possible due to the unsuitability of the terrain as well as the high cost. Because the units were small, it was decided to allow the turbine to achieve runaway speed (in the event of a load rejection) before the gates would begin to close off the waterflow. It is a characteristic of Francis turbines to pass less water at runaway than at synchronous speed. Axial flow (Kaplan) turbines however do not exhibit this characteristic. The unit discharge for the Francis turbine decreases with increasing PHI. Thus, pressure rises due to water hammer will always occur in systems with Francis turbines regardless of the gate closure rate. The minimum pressure rise is that which will occur when the unit accelerates to runaway while the gates remain fixed. The rate of acceleration directly influences the magnitude of the pressure rise and depends only upon the turbine characteristics and the inertia

(WR^2) of the rotating parts. Therefore, by adding inertia in the form of a flywheel, the pressure rise can be diminished. For small turbine-generator units such as these, it is technically and economically feasible to do this.

The Ponape project will consist of a new powerhouse with a 4600 ft (modeled) long penstock and two different sizes of horizontal Francis turbine-generator units both rated at a net head of 200 ft. A recent Francis turbine model was used for this analysis. Prototype runner diameters of 1.33 ft and 1.875 ft and synchronous speeds of 1200 rpm and 900 rpm respectively, were predicted by this model. The large turbine was "rated" at approximately 1070 kW and the small turbine at 530 kW to produce the guaranteed plant electrical output of 1600 kW rated at the net head of 200 feet. The small unit has minimum tailwater and forebay elevations of 139.5 and 362.5 ft respectively, with the elevations of the larger turbine being one-half foot higher in both instances.

The purpose of running the WHAMO program for the Ponape Project was to model hydraulic transient (water hammer) pressure rise due to plant load rejection and subsequent unit runaway. The water hammer phenomenon arises wherever a change in flow (Q) occurs causing positive pressure waves for decrease in Q and negative waves for increases in Q. If the initiation of gate closure is delayed until after the unit is at runaway speed, the magnitude of the hydraulic transient pressures will be minimized. This was the approach taken. The penstock design pressure was set at 125 psi (130 percent of maximum static head at the powerhouse). The Ponape system was modeled for the worst foreseen conditions of simultaneous full gate load rejection with 1600 kW plant output. The gates would then be closed at such a rate as to not exceed the penstock design pressure.

The Ponape system construction in terms of the WHAMO program is detailed in Figure 28. The penstock element arrangement consists of one variable diameter conduit, seven constant diameter conduits, and one branch, the total of which connects the forebay to the turbine. The draft tubes are modeled as well. The elevation of each node is shown with the headwater being at El. 362.4 and the tailwater being at El. 140.4. The conduit was broken into several individual elements at every change in property or slope, with a node connecting each. The estimated diameters and developed lengths of the penstock segment are also shown. The penstock is of steel reinforced concrete and contains no surge tank.

The WHAMO input data served to link the elements in the Ponape System together in the appropriate order. Elevations, lengths, areas, celerity, friction coefficients, windage and turbine speed, diameter, and WR^2 were all entered data. Twelve gate

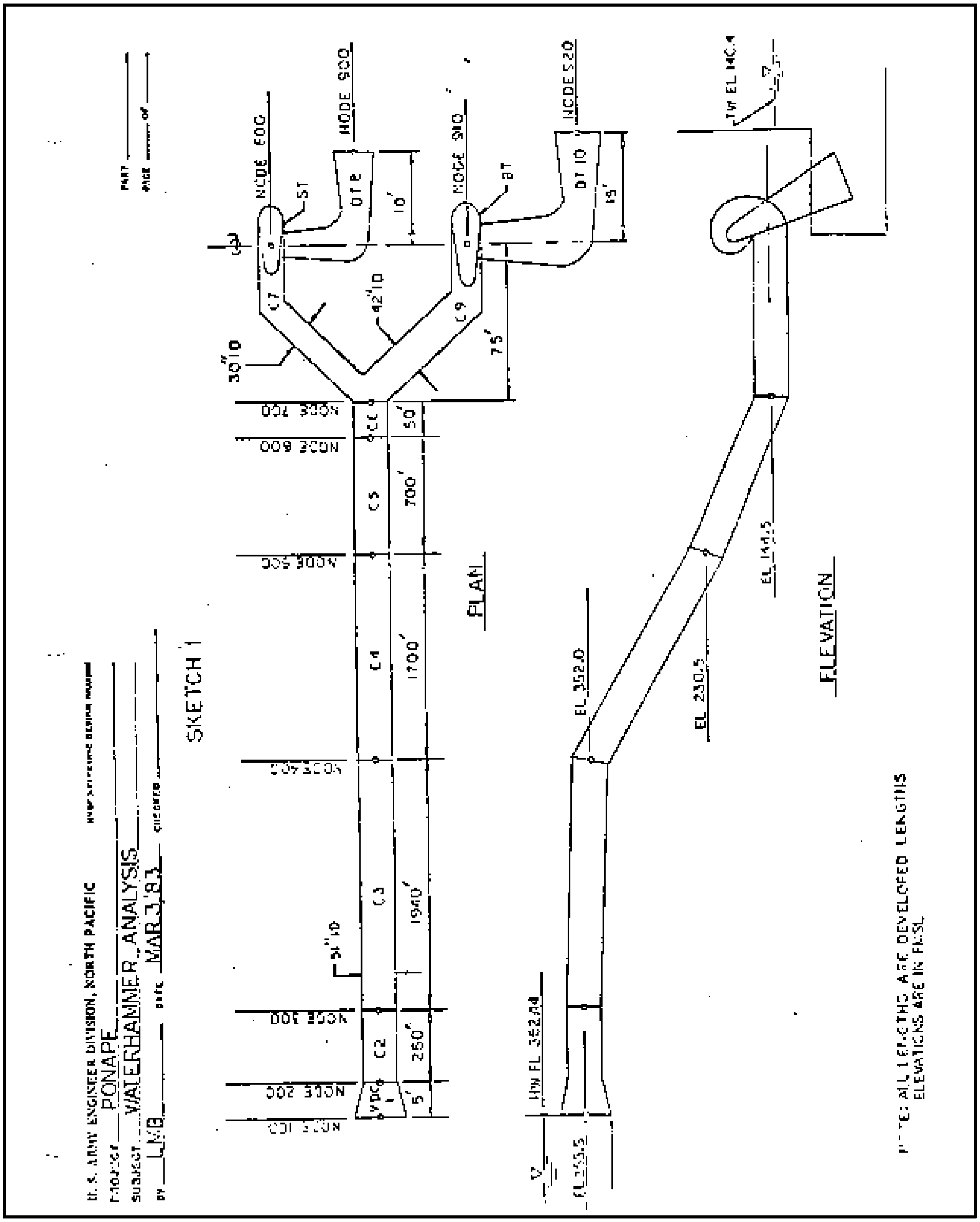


Figure 28. Ponape system schematic.

and twenty-four phi (velocity ratio) values were also entered, forming the basis of the two 240 element arrays of turbine model data, one for horsepower and the other for discharge. These values were digitized directly from existing model curves and are displayed as a table in the output generated under Turbine Characteristics.

WR^2 (moment of inertia of rotating parts) was increased over several runs in order to delay maximum pressure occurrence. The design energy elevation at the powerhouse was determined to be 428.15 fmsl (125 psi) and corresponds to the maximum pressure the penstock will be designed to withstand. The results of the pertinent WHAMO runs are as follows:

Run #	Maximum Energy Elevation	WR^2 Unit 1	Unit 2	Time After Load Rejection (sec)
1	445.6	1060	3830	2.75
2	413.6	2120	7660	4.0
3	400.5	3180	11490	4.75
4	418.4	1820	7000	3.75

Run #4 was the final run, with the WR^2 chosen to keep the energy elevation below the 428.15 fmsl design condition. The results of this run yielded a maximum pressure surge of 418.4 fmsl (120 psi) at the turbine under full gate conditions.

Shown are computer generated plots displaying the time varying energy elevation, discharge, speed, and power of each turbine (Figures 29-32). The maximum energy elevation grade line along the length of the penstock is shown on Figure 33. The input file is shown in Figure 34.

Immediately after reaching maximum runaway speed, the calculated Q (discharge) and HP (horsepower) values exceed the range of turbine model data. However, since this error occurs after the maximum hydraulic transient pressure and runaway speed are attained, the computer results are accurate. A second hydraulic transient occurs when the gates begin to close. The magnitude of this transient is directly influenced by the gate closure rate. Although this could be modeled, there is no need since the gate timing rate can simply be adjusted in the field by doing load rejection tests. The closure setting should be one that produces a maximum pressure of 125 psi in the penstock at the powerhouse.

It should also be noted that the turbine model used by the manufacturer will not be the same as that used in this study. As explained previously the turbine "chokes" the flow of water passing through it as it accelerates. The amount of "choking" is related to the specific speed of the turbine. Generally speaking, the higher the specific speed the more "choking" occurs during runaway. The model specific speed is quite close to the proposed turbines' specific speeds (at their rated points). Therefore the results obtained from this study should be representative even though the characteristics of the turbine actually furnished could not be simulated.

The penstock and turbine water passages will be hydrostatically tested to 150% of their pressure. Also the turbine generator units will be designed and tested to be

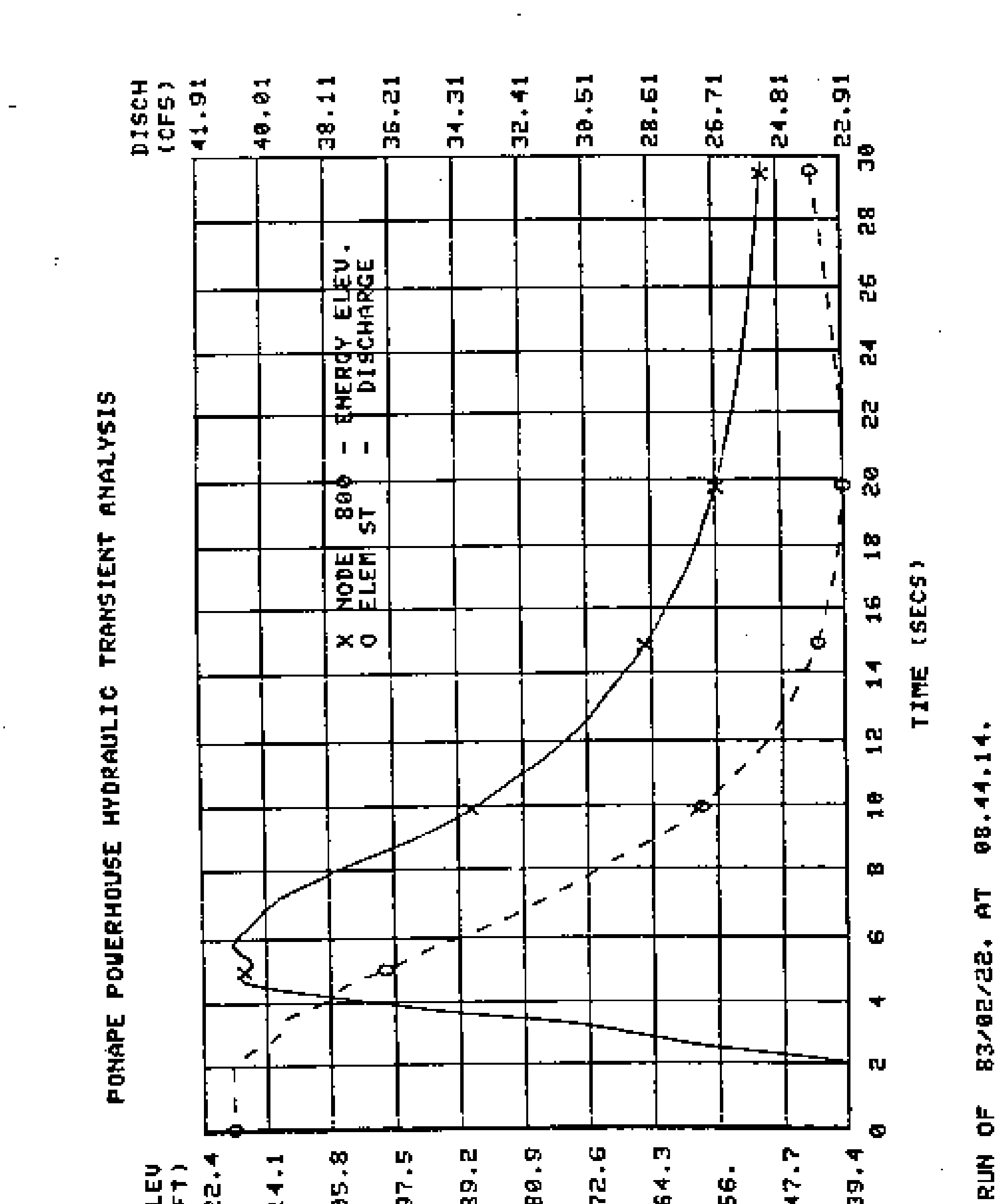


Figure 29. Simulation results for small turbine unit (Run #4).

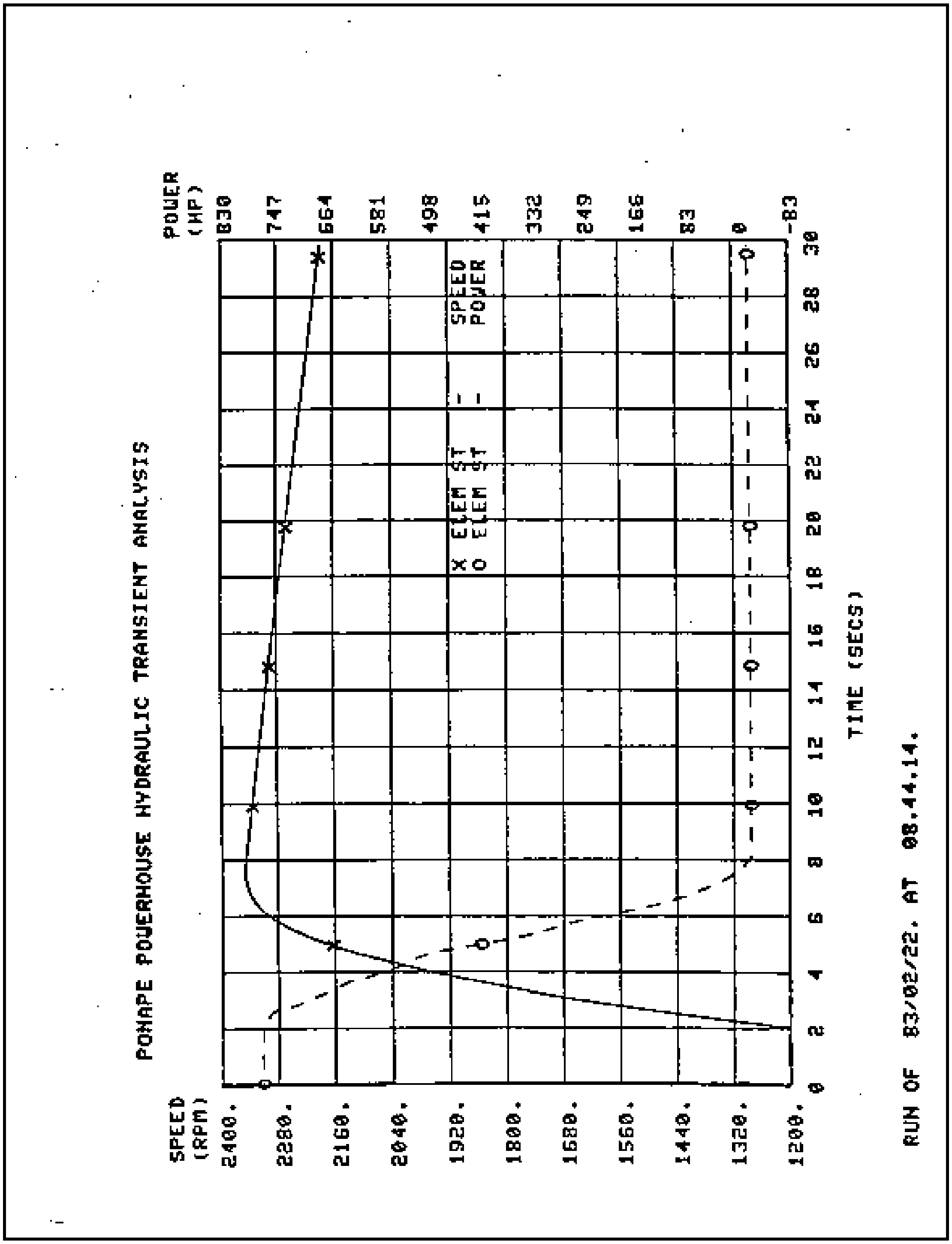


Figure 30. Simulation results for small turbine unit (Run #4).

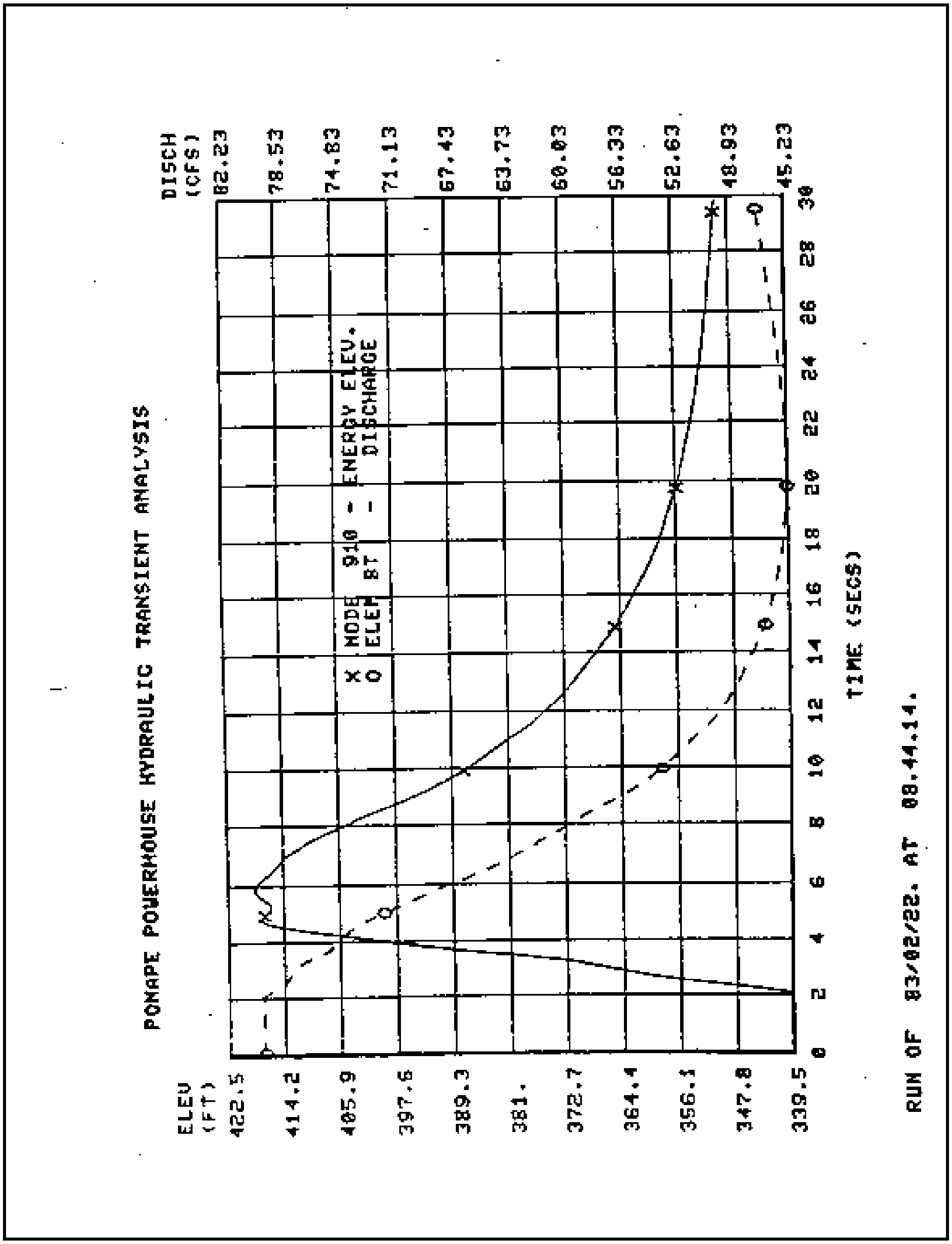


Figure 31. Simulation results for big turbine unit (Run #4).

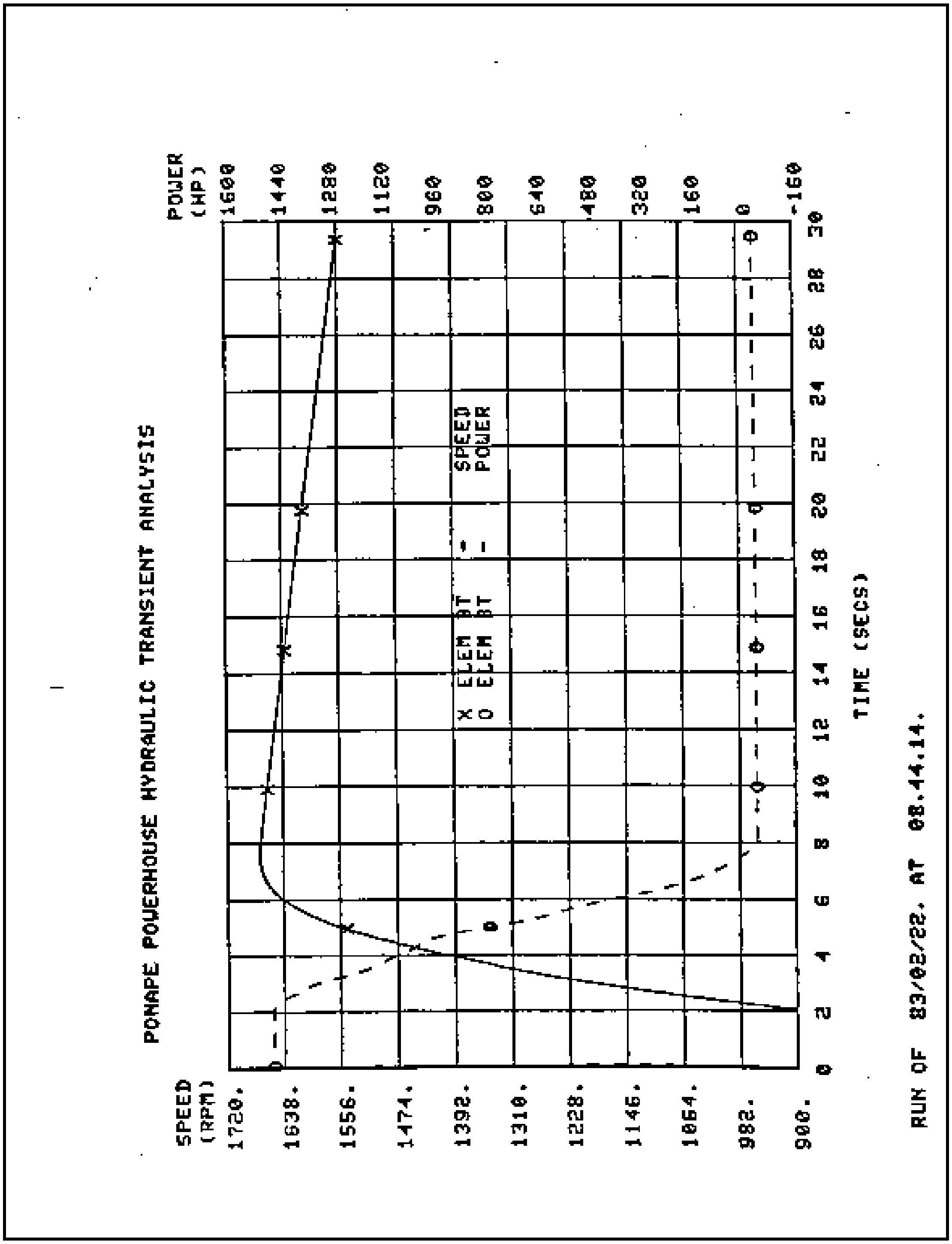


Figure 32. Simulation results for big turbine unit (Run #4).

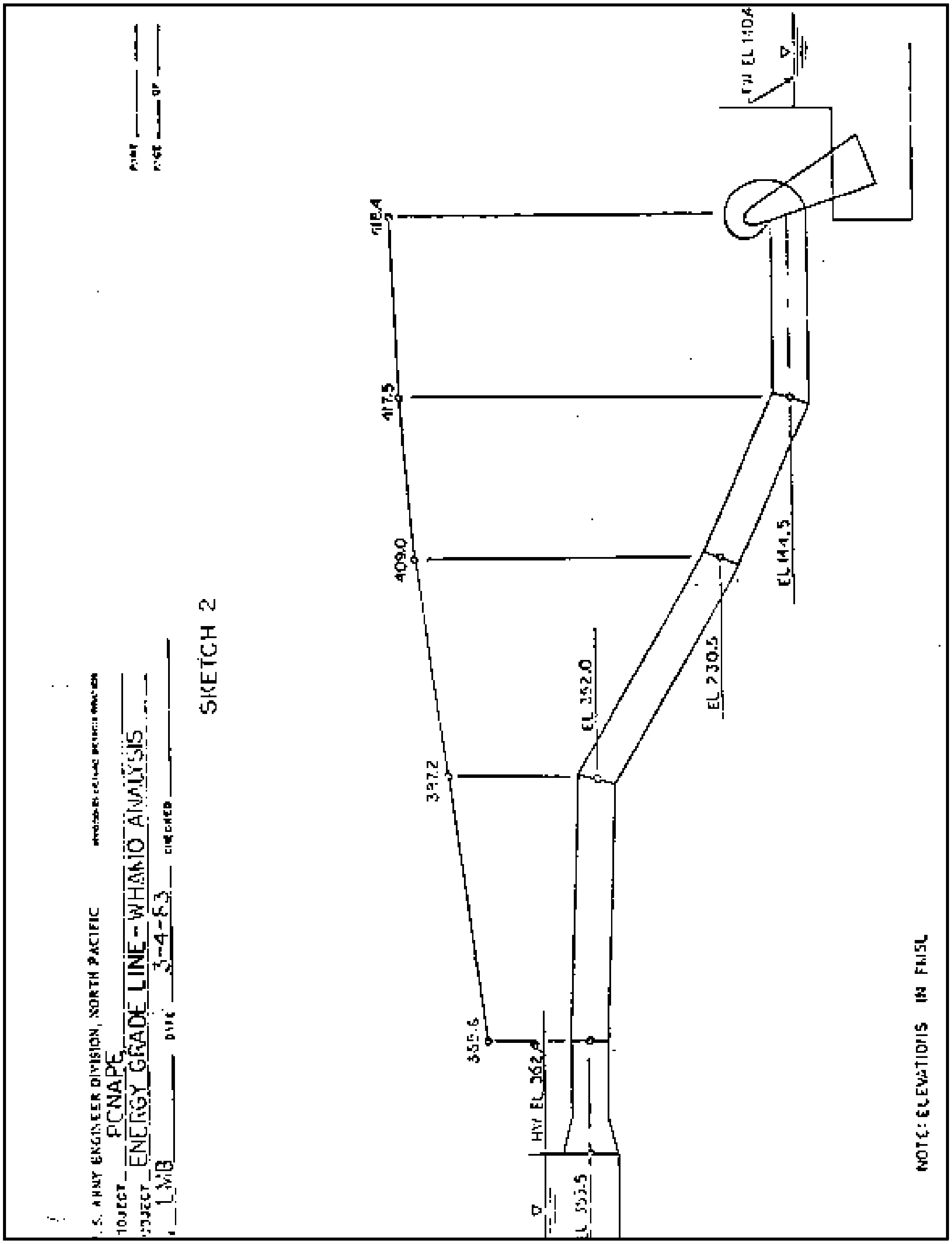


Figure 33. Maximum energy grade line (Run #4).

```
PONAPE POWER PLANT
SYSTEM
ELEM HW AT 100

ELEM VDC1 LINK 100 200
ELEM C2 LINK 200 300
ELEM C3 LINK 300 400
ELEM C4 LINK 400 500
ELEM C5 LINK 500 600
ELEM C6 LINK 600 700
JUNCTION AT 700
ELEM C7 LINK 700 800
ELEM ST LINK 800 850                    .
ELEM DTS LINK 850 900
ELEM TW AT 900
ELEM C9 LINK 700 910
ELEM BT LINK 910 915
ELEM DT10 LINK 915 920
ELEM TW AT 920
NODE 300 ELEV 355.5
MUDE 400 ELEV 352.0
HDDE 500 ELEV 230.5

NODE 600 ELEV 144.5
NCLE 800 ELEV 144.5
NODE 910 ELEV 144.5
FINISH

RESERVOIR ID HW ELEV 362.44 FINISH

COND ID VDC1 VARIABLE DIST 0.0 AREA 22. DIST 5. AREA 14.19 LENgTH 5.
            CELERITY 3600. FRICTION .0185 FINISH
COND ID C2 LENGTH 260. DIAM 4.25 CELER 36OO. FRICT .0185  FINI
COND ID C3 LENG 1940. DIAM 4.25 CELER 35OO. FRICT .0185 NUMSEG 5 FINI
COND ID C4 LENG 1700. DIAM 4.25 CELER 36OO. FRICT .0185 NUMSEG 5  FINI
COND ID C5 LENG 700. DIAM 4.25 CELER 36OO. FRICT .0185  FINI
COND ID C6 LENG 50. DIAM 4.25 CELER 36OO. FRICT .0185 NUMSEG 2  FINI
COND ID C7 LENGTH 25. DIAM 2.5 CELER 36OO. FRICT .0185  FINI
TURBINE ID ST TYPE 1 SYNCSPEED 1200. DIAM 1.333 WINDAGE 3.5 FRICT1ON
                  3.5 WR2 1820. FINI
COND ID DT8 VARIABLE DIST 0.0 AREA 1.396 DIST 10. AREA 3.67
            LENG 10. CELER 3600. FRICT .Ol85 FlNI
COND ID C9 LENG 25. DIAM 3.5 CELER 3600. FRICT .0185 FlNI
TURBINE ID BT TYPE 1 SYNCSPEED 900. DIAM 1.875 WINDAGE 7.5
               FRICTION 7.5 WR2 7000. FINI
COND ID DT10 VARIABLE DIST 0.0 AREA 2.76 DIST 15. AREA 7.33
           LENGTH 15. CELER 3600. FRICT .0185 FINI

RESERVOIR ID TW ELEV 140.4 FINI

TCHARACTERISTICS TYPE 1


  (TURBINE CHARACTERISTICS NOT SHOWN.  PROPRIETARY INFORMATION)
```

Figure 34. Ponape input file.

```
DISPLAY ALL FINI
HISTORY
NODE 300 HEAD
NODE 400 HEAD
NODE 500 HEAD
NODE 600 HEAD
NODE 800 HEAD Q

ELEM ST SPEED Q POWER POSITION HEAD
ELEM PT SPEED Q POWER HEAD
FINISH
SNAPSHOT TIME 7.0 FIE1SH
PLOTFILE
NODE 300 HEAD
NODE 400 HEAD
NODE 500 HEAD
NODE 600 HEAD
NODE 800 HEAD Q
NODE 910 HEAD Q
ELEM ST SPEED Q POWER POSITION HEAD
ELEM PT SPEED Q POWER HEAD
FINISH

CONTROL
BTCOMP 0.05
DTOUT 0.25
TMAX 20.0
DTCOMP 0.10
DTOUT 1.0
TMAX 50.
FINISH

OPTURBINE ID ST REJECT TOFF 2.0 GSCHED 1   FINI
OPTURBINE ID BT REJECT TOFF 2.0 GSCHED 1 FINI

SCHEDULE GSCHEDULE 1

T 0.0 G 100.
T 5.0 G 100.
T 10.0 G 100.
T 20. G 100.
T 30. G 100.
T 32. G 90.
T 35. G 75.
T 40. G 50.
1 45. G 25.
T 50. G 0.
FINISH
GO
GOODBYE
END OF DATA
```

Figure 34 (continued)

capable of operating at runaway speed safely and without damage. Load rejection tests will need to be performed in the field to verify the predicted performance and set the gate closure rate.

The maximum transient pressure determined by the WHAMO model of the Ponape System is 418.48 fmsl (120 psi) at the powerhouse which is less than the penstock design pressure (125 psi). This corresponds to maximum runaway speeds of 2350 rpm and 1675 rpm for the small and large turbines respectively. Minimum values of WR^2 of the rotating parts of the small and large turbines should be specified to be 1820 lb-ft^2 and 7000 lb-ft^2 respectively.

6 Basic Algorithms

The basic algorithms used to model the hydraulics and mechanics of the WHAMO elements, along with those used to model the interrelationship of these elements in an integrated system, are presented in this chapter. The presentation does not include the theory and technical background of these algorithms. Nor are the hydraulic or mathematical derivations generally included. These can be found in the publications referenced in this chapter and elsewhere in the report.

6.1 Governing Equations

Governing equations, which use the momentum or energy conservation and continuity principles to model the hydraulics of an element, have been introduced in Chapter 2. The form of these equations for the different element types is summarized below and elaborated upon in the subsections which follow.

<u>Conduit Equations</u>

Momentum:

$$\frac{1}{gA}\frac{\partial Q}{\partial t} + \frac{\partial H}{\partial x} + \frac{f}{D}\frac{Q|Q|}{2gA^2} = 0$$

Continuity:

$$\frac{\partial H}{\partial t} + \frac{c^2}{gA}\frac{\partial Q}{\partial x} = 0$$

<u>Junction Equations</u>

Energy:

$$H_i = H_j - loss_{ij}$$

$$H_i = H_k - loss_{ik} \quad etc.$$

Continuity:

$$Q_i + Q_j + Q_k + \ldots = 0$$

Machine and Valve Equations

Energy: $f(H_i - H_j, Q) = 0$

Continuity: $Q_i = Q_j$

Reservoir Boundary Equation

Energy: $H_i = HRES - loss$

Flow Boundary Equation

Continuity: $Q_i = QBC$

Surge Tank Free Surface Equation (no outflow)

Continuity: $$\frac{dH}{dt} - \frac{Q}{A}\left[\frac{1}{g}\frac{dv}{dt} + 1\right] = 0$$

Air Chamber Boundary Equation (unvented)

$$\frac{dH}{dt} = \left[\frac{n}{v}\left(H - \frac{v^2}{2g} - WS + HB\right) + \frac{1}{A} + \frac{1}{gA}\frac{dv}{dt}\right]Q$$

In order to be utilized in the computational scheme of the model, the governing equations must be approximated (where necessary) to a linear, algebraic form. The linearization and finite difference techniques are discussed in the subsequent subsections. The set of linear, finite difference equations which model the hydraulics of all elements of a system are termed the system inner equations.

6.2 Unsteady, Compressible Flow in Elastic Conduits

The form of the partial differential equations of motion (momentum and continuity) used by this model to represent unsteady, compressible flow in elastic conduits is presented. This is followed by the general finite difference representation of the differential equations adopted for WHAMO. The difference scheme is then applied to the momentum and continuity equations. The result is altered slightly to a linear form suitable for simultaneous solution with the equations for other system components. A more complete treatment, including derivations, can be found in Resource Analysis, Inc. (1976).

6.2.1 Momentum and Continuity Equations

The simplified, one-dimensional continuity and momentum equations utilized in WHAMO model development are as shown. Note that these use total head, H, and discharge, Q, as dependent variables and that fluid compressibility and conduit elasticity are implicit in the celerity term, c.

Momentum:

$$\frac{1}{gA}\frac{\partial Q}{\partial t} + \frac{\partial H}{\partial x} + \frac{f}{D}\frac{Q|Q|}{2gA^2} = 0 \qquad \textbf{(6.1)}$$

Continuity:

$$\frac{\partial H}{\partial t} + \frac{c^2}{gA}\frac{\partial Q}{\partial x} = 0 \qquad \textbf{(6.2)}$$

where,

- H = Total head or energy grade
- Q = discharge
- x = distance along the conduit
- t = time
- g = gravitational constant
- A = cross sectional area of the conduit
- D = diameter of the conduit
- f = Darcy-Weisbach friction factor
- c = celerity of a compression wave traveling through this conduit (see Section 3.3.2)

6.2.2 Finite Difference Scheme

The WHAMO program is formulated in terms of a four-point implicit finite difference representation of the governing differential equations. This implicit scheme can be summarized by noting how it is applied to an arbitrary function of space and time, f(x,t), and its derivatives. Thus, using the notation:

$$f(j\Delta x, n\Delta t) = f_j^n$$

the general finite difference scheme is:

$$f(x,t) = \left[\frac{f_{j+1}^{n+1} + f_j^{n+1}}{2} \right] + (1-\theta) \left[\frac{f_{j+1}^{n} + f_j^{n}}{2} \right] \quad \textbf{(6.3a)}$$

$$\frac{\partial f\ (x,t)}{\partial t} = \frac{1}{2} \left[\frac{f_{j+1}^{n+1} - f_{j+1}^{n}}{\Delta t} + \frac{f_j^{n+1} - f_j^{n}}{\Delta t} \right] \quad \textbf{(6.3b)}$$

$$\frac{\partial f\ (x,t)}{\partial x} = \theta \left[\frac{f_{j+1}^{n+1} - f_j^{n+1}}{\Delta x} \right] + (1-\theta) \left[\frac{f_{j+1}^{n} - f_j^{n}}{\Delta x} \right] \quad \textbf{(6.3c)}$$

where ⊖ is a weighting factor included for numerical stability.

This general representation is applied to the momentum and continuity equations in conduits, to the surge tank free surface equation, and to other differential equations which govern model algorithms.

6.2.3 Momentum and Continuity in Finite Difference Form

The momentum and continuity equations (6.1, 6.2) can now be represented in algebraic, finite difference form as follows:

Momentum:

$$- H_j^{n+1} + \left[\frac{\Delta x_j}{2g\theta A_j \Delta t} \right] Q_j^{n+1} + H_{j+1}^{n+1} + \left[\frac{\Delta x_j}{2g\theta A_j \Delta t} \right] Q_{J+1}^{n+1} \quad \textbf{(6.4)}$$

$$= \frac{1-\theta}{\theta}\left[H_j^n - H_{j+1}^n \right] + \left[\frac{\Delta x_j}{2g\theta A_j \Delta t} \right] \left[Q_j^n + Q_{j+1}^n \right]$$

$$- \frac{\Delta x_j f_j}{4g\theta D_j A_j^2} \left[Q_j^n \mid Q_j^n \mid + Q_{j+1}^n \mid Q_{j+1}^n \mid \right]$$

Valid only for $\Theta \neq 0$

Continuity:

$$H_j^{n+1} - \left[\frac{2\theta c_j^2 \Delta t}{gA_j \Delta x_j} \right] Q_j^{n+1} + H_{j+1}^n + \left[\frac{2\theta c_j^2 \Delta t}{gA_j \Delta x_j} \right] Q_{j+1}^{n+1} \quad \textbf{(6.5)}$$

$$= (H_j^n + H_{j+1}^n) + \left[\frac{2(1-\theta) c_j^2 \Delta t}{gA_j \Delta x_j} \right] \left[Q_j^n - Q_{j+1}^n \right]$$

The above equations are faithful to the general finite difference scheme in all but one aspect. The non-linear Q|Q| term of the momentum equation (6.1) is represented using only Q at the previous time step, instead of using weighted averages of Q at the previous and future time steps. As a result, the finite difference momentum equation (6.4), as well as the finite difference continuity equation (6.5), is completely linear in terms of the unknown dependent variables,

$$H_j^{n+1}, \quad H_{j+1}^{n+1}, \; Q_j^{n+1} \quad and \quad Q_{j+1}^{n+1}.$$

(Note that the solution proceeds from time step to time step so that H and Q at the previous time step are always known constant quantities.)

Equations (6.4) and (6.5) can be simplified in appearance by introducing the following coefficients:

$$\alpha_j = \frac{2\theta c_j^2 \Delta t}{gA_j \Delta x_j} \quad \textbf{(6.6a)}$$

$$\beta_j = (H_j^n + H_{j+1}^n) + (\frac{1-\theta}{\theta})\ \alpha_j\ (Q_j^n - Q_{j+1}^n) \qquad \textbf{(6.6b)}$$

$$\gamma_j = \frac{\Delta x_j}{2g\theta A_j \Delta t} \qquad \textbf{(6.6c)}$$

$$\delta_j = (\frac{1-\theta}{\theta})\ (H_j^n - H_{j+1}^n) + \gamma_j\ (Q_j^n + Q_{j+1}^n) - \qquad \textbf{(6.6d)}$$

$$\frac{\Delta x_j\ f_j}{4g\theta D_j A_j^2} \left[Q_j^n \mid Q_j^n \mid + Q_{j+1}^n \mid Q_{j+1}^n \right]$$

Note that the coefficients, α_j and Υ_j, are not functions of time and thus need to be evaluated only once for each section. The remaining coefficients do vary with time and must therefore be re-evaluated each time the solution is advanced to a new time row.

In terms of these coefficients, the numerical equations for the j^th segment become:

Momentum: $$-H_j^{n+1} + \gamma_j\ Q_j^{n+1} + H_{j+1}^{n+1} + \gamma_j\ Q_{j+1}^{n+1} = \delta_j \qquad \textbf{(6.7)}$$

Continuity: $$H_j^{n+1} - \alpha_j\ Q_j^{n+1} + H_{j+1}^{n+1} + \alpha_j\ Q_{j+1}^{n+1} = \beta_j \qquad \textbf{(6.8)}$$

A pair of equations of this form applies to every segment of a conduit in a system modeled by WHAMO.

Discussion of the use of finite difference techniques in the solution of differential equations can be found in Perkins (1964) Part II and in Carnahan (1969).

6.3 Algorithms for Flow Elements

6.3.1 Prismatic Conduits

The governing equations for conduits are the continuity and momentum equations introduced in Section 6.2. In linearized finite difference form these are equations (6.7) and (6.8).

An important term in these equations is the celerity, c. WHAMO allows the celerity of a conduit to be specified as input or, alternatively, it will be computed by the program from the conduit properties. The following formula is used by the program to compute celerity.

$$c = \frac{4720}{\sqrt{1 + (3 \cdot 10^5/E) \cdot (D/WT)}} \quad \textbf{(6.9)}$$

where,

c	=	wave celerity
E	=	modulus of elasticity of pipe wall
WT	=	thickness of pipe wall
D	=	diameter of the conduit

Local head losses in conduits may be accounted for by incorporating an extra term in the momentum equation for segments where an added loss occurs. This term takes the form

$$C_{add} \frac{Q|Q|}{2gA^2}$$

The coefficient, C_{add}, may in general be a function of the flow direction. The friction and added loss terms in the momentum equation then combine to give the total loss as:

$$Total\ Loss\ Term = \frac{f\Delta x}{D} \frac{Q|Q|}{2gA^2} + C_{add} \frac{Q|Q|}{2gA^2}$$

$$= \left[\frac{f\Delta x}{D} + C_{add} \right] \frac{Q|Q|}{2gA^2}$$

$$= \frac{f^1}{D} \frac{Q|Q|}{2gA^2}$$

in which f^1 is an equivalent friction factor designed to satisfy the above relations. In all that follows the momentum equation will be written in terms of this equivalent friction factor. As far as the program user is concerned, however, separate values of f and c_{add} will be required to be specified. For the sake of clarity, the equivalent friction factor will be represented by f.

Entrance and exit losses at the boundary between a conduit and a reservoir are calculated using the same formula:

$$H.L. = C_{add} \frac{Q|Q|}{2gA^2} \tag{6.10}$$

Instead of including this term in the momentum equation of a conduit segment, however, end loss is subtracted from the specified head value at the reservoir.

6.3.2 Variable Diameter Conduits

The following derivation shows that the continuity equation presented in Section 6.2 applies to variable diameter conduits as well as to prismatic conduits. This derivation is included because it does not appear elsewhere.

The continuity equation for non-constant diameter conduits given by Wylie (1978) is reproduced below adopting the notation of this manual and ignoring lateral inflows.

$$\frac{g}{C^2}\left[v \frac{\partial h}{\partial x} + \frac{\partial h}{\partial t}\right] + \frac{\partial v}{\partial x} + \frac{2v}{D}\frac{dD}{dx} = 0 \tag{6.11}$$

The term v represents velocity, h is piezometric head, and $\frac{dD}{dx}$ is the rate of expansion (or contraction) of the conduit where v << c, the convective acceleration, is usually ignored (Wylie, p. 306). Additionally, as shown in Resource Analysis, Inc. (1976), $\frac{\partial h}{\partial t}$ can be replaced by $\frac{\partial H}{\partial t}$ with the result:

$$\frac{\partial H}{\partial t} + \frac{c^2}{g}\frac{\partial v}{\partial x} + \frac{c^2}{g}\frac{2v}{D}\frac{dD}{dx} = 0 \qquad \textbf{(6.12)}$$

Now replace v with Q/A to give:

$$\frac{\partial H}{\partial t} + \frac{c^2}{g}\frac{\partial (Q/A)}{\partial x} + \frac{c^2}{g}\frac{2Q}{DA}\frac{dD}{dx} = 0 \qquad \textbf{(6.13)}$$

Replacing:

$$\frac{\partial (Q/A)}{\partial x} = \frac{1}{A}\frac{\partial Q}{\partial x} - \frac{Q}{A^2}\frac{dA}{dx} \qquad \textbf{(6.14)}$$

$$= \frac{1}{A}\frac{\partial Q}{\partial x} - \frac{Q}{A^2}\left[\frac{\pi D}{2}\frac{dD}{dx}\right]$$

in the equation (6.13) gives:

$$\frac{\partial H}{\partial t} + \frac{c^2}{g}\left[\frac{1}{A}\frac{\partial Q}{\partial x} - \frac{Q}{A^2}\frac{\pi D}{2}\frac{dD}{dx}\right] + \frac{c^2}{g}\frac{2}{D}\frac{Q}{A}\frac{dD}{dx} = 0 \qquad \textbf{(6.15)}$$

Observe that:

$$\frac{c^2}{g}\frac{Q}{A^2}\frac{\pi D}{2}\frac{dD}{dx} = \frac{c^2}{g}\frac{2}{D}\frac{Q}{A}\frac{dD}{dx}$$

so the simplified form of the continuity equation is

$$\frac{\partial H}{\partial t} + \frac{c(x)^2}{gA(x)}\frac{\partial Q}{\partial x} = 0$$

This form is unchanged from equation (6.2) except for the emphasis that A and C may be functions of distance along a pipe. Evidently, writing the equation in terms of Q rather than v implicitly allows for a non-constant diameter.

The momentum equation given by Wylie (1978) is identical for prismatic conduits and variable diameter conduits, so that the momentum equation (6.1) in Section 6.2 also applies to variable diameter conduits.

As the governing differential equations of Section 6.2 apply to variable diameter conduits, so do the linearized, finite difference equations (6.7) and (6.8). The only difference is that D, A, and c may vary along a conduit. For each segment of a conduit, values of α, Υ, are used which are the average of the values computed using the properties at the ends of the segment. Similarly, the term $\Delta xf/(4g\ominus DA^2)$ used to compute δ for a segment in equation (6.6d) is an average of values computed at the ends of the segment.

6.3.3 Simple Valves

The calculation of flow through a valve is based on the formula

$$Q = C_Q D^2 \sqrt{g} \sqrt{\Delta H} \qquad \textbf{(6.16a)}$$

used in Waterways Experiment Station (1977) for butterfly valves. The term C_Q is a discharge coefficient which is a function of valve position and flow direction, D is the valve diameter, and ΔH is the head loss across the valve.

The formula can be rearranged to:

$$H_j - H_{j+1} = \left[\frac{1}{C_Q^2 D^4 g} \right] | Q_j | Q_j \qquad \textbf{(6.16b)}$$

and then linearized to the form

$$H_j - H_{j+1} = SLOPE \cdot Q_j + CONST \qquad \textbf{(6.17)}$$

The slope and intercept terms are computed from the derivative of equation (6.16) and the computed ΔH using an estimated value of the discharge, $Q_{est'}$ equal to the discharge at the previous time step.

$$SLOPE = 2\left[\frac{1}{C_Q^2 D^4 g}\right] \left| Q_{est} \right| \quad \textbf{(6.17a)}$$

$$CONST = -\left[\frac{1}{C_Q^2 D^4 g}\right] Q_{est} \left| Q_{est} \right| \quad \textbf{(6.17b)}$$

Equation (6.17) above is the energy equation used in modeling a valve. The continuity equation is simply

$$Q_{j+1} = Q_j \tag{6.18}$$

In order to model a valve, the coefficient C_Q must be known at every time step. The user must therefore input a set of valve characteristics, C_Q versus valve position and a schedule of valve position versus simulation time. Note that different sets of characteristics may apply to forward and reverse flow through the valve. As an alternative to specifying the valve characteristics in terms of the discharge coefficient, C_Q, in equation (6.16), the user may, if more convenient, specify the characteristics in terms of a head loss coefficient, C_H, defined

$$C_H = \frac{\Delta H}{V^2/2g} \tag{6.19}$$

The conversion from C_H to C_Q is simply

$$C_Q = \frac{\pi}{\sqrt{8}\,\sqrt{C_H}} \tag{6.20}$$

Default characteristics for butterfly valves, gate valves, Howell-Bunger valves, and spherical valves have been written into the program. The former were taken from Waterways Experiment Station (1977), charts 331-1 and 330-1, respectively. The spherical valve characteristics were taken from a curve developed by Boving and Co., Ltd., for the "Havasu rotary valve." The discharge coefficient for Howell-Bunger valves are assumed linear with position and equal to 0.92 at the full open position.

6.3.4 Non-Return Valves

Equations (6.16-6.20) are applied to flow through non-return valves as well as other valve types. For non-return valves, a single user-specified discharge or head loss coefficient applies to forward flow, and an extremely small C_Q is used in the reverse direction to essentially shut off reverse flow.

At any time step that flow through a ONEWAY valve element is seen to change to reverse direction, the computation is repeated and the time step length shortened to the point that reverse flow is negligible. This is a simple representation of a check valve. No significant reverse flow is allowed, so check valve "slamming" which is sometimes experienced, and associated transients, are not modeled.

6.3.5 Pressure Actuated Valves

For RELIEF elements, valve position is specified not as a function of time, but as a function of pressure upstream (or at some other reference point). During simulation, the computed reference pressure at the previous time step is used to estimate current valve position. Computations can then be made as for simple valves using equations (6.16-6.20). If the newly computed reference pressure corresponds with a valve position significantly different from the estimated value, then the computations are repeated at a time step short enough that the difference between computed and estimated valve position is within a user specified tolerance.

6.3.6 Expansions and Contractions

The diameter change element (Figure 35) simply subtracts energy from the system at the point specified. The applicable energy equation is

$$H_{j+1} = H_j - H.L. \tag{6.21}$$

where H.L. is the calculated head loss. The continuity equation is again

$$Q_{j+1} = Q_j$$

The head loss term is calculated using an estimated discharge equal to the discharge at the previous time step.

$$H.L. = C \frac{Q_{est}^2}{2A^2 g} \tag{6.22}$$

The head loss coefficient, C, will in general be different depending on whether flow is in the expanding direction or the contracting direction.

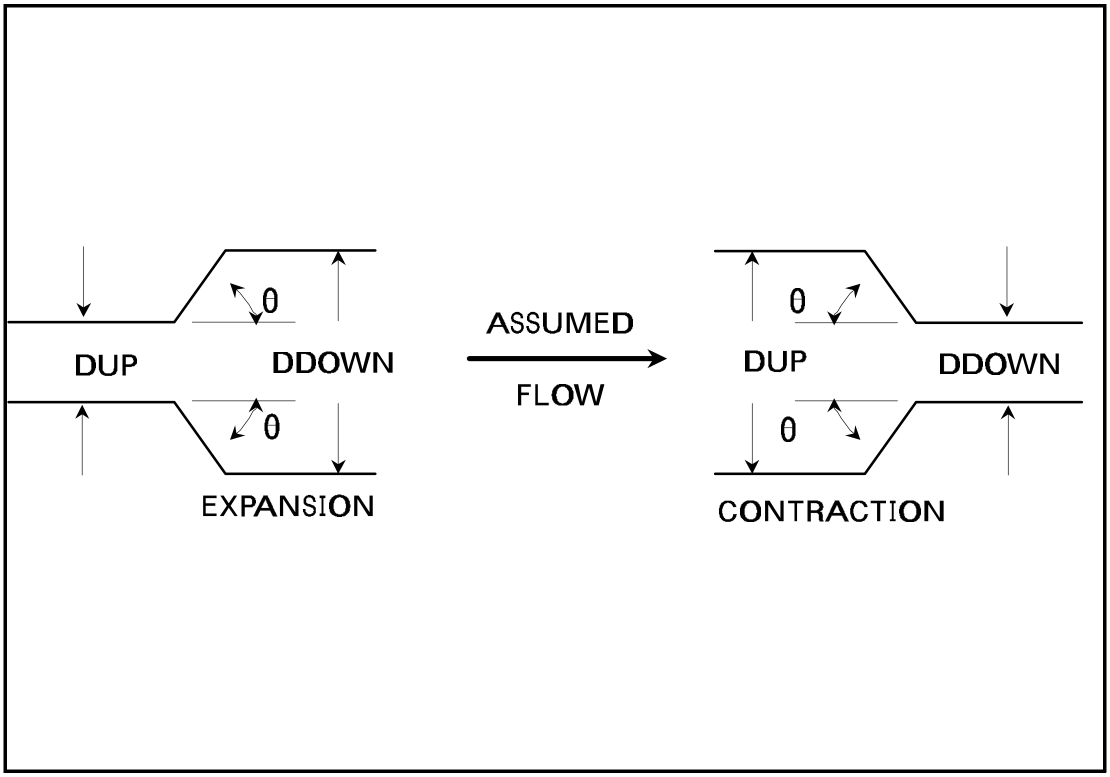


Figure 35. Definition sketch for diameter change elements.

The value of the head loss coefficients may be specified as input or they may be calculated from the geometry by the program. The following formulae are used in this calculation. These are taken from Crane (1979).

For $\theta \leq 22.5°$: $$C_e = 2.6 \sin\theta \ (1-\beta^2)^2 \qquad \textbf{(6.23a)}$$

$$C_c = 0.8 \sin\theta \ (1-\beta^2) \qquad \textbf{(6.23b)}$$

For $\theta > 22.5°$: $$C_e = (1-\beta^2)^2 \qquad \textbf{(6.23c)}$$

$$C_c = 0.5 \ (1-\beta^2) \sqrt{\sin\theta} \qquad \textbf{(6.23d)}$$

where,

C_e is the coefficient for expansion
C_c is the coefficient for contraction

$$\beta = \frac{D_1}{D_2}$$

D_1 is the smaller of DUP and DDOWN,
D_2 is the larger of DUP and DDOWN, and
DUP, DDOWN and $\ominus$ are shown in Figure 35.

6.4 Algorithms for Turbo-machines

The continuity-energy equation pair for pumps, turbines, and pump turbines take the following form.

Continuity: $$Q_{j+1} = Q_j$$ (see Eq 6.18)

Energy: $$\Delta H = SLOPE \cdot Q_j + CONST$$ (see Eq 6.17)

where, $$\Delta H = H_{j+1} - H_j \quad : \quad turbines + pump-turbines$$

$$\Delta H = H_j - H_{j+1} \quad : \quad pumps$$

The energy equation, $\Delta H = f(Q)$, is derived from a set of machine characteristics which is supplied as input. These characteristics relate, in some manner, the discharge, Q, the head difference, ΔH, the rotational speed, ω, and, for turbines and pump-turbines, the wicket gate position, G. The exact form of the characteristics is different for the different types of turbo-machine. In general, the characteristics are expressed using dimensionless or model scale values.

The determination of a linear ΔH vs. Q energy equation from the machine characteristics at any time step follows this general procedure:

(1) The operating point in the table is estimated. The wicket gate position is determined from the gate schedule or from the governor equation. The estimated speed and discharge are extrapolated from the values at the previous 2 time steps. These variables, rather than ΔH, are extrapolated because their behavior in a transient state is much more regular.

(2) The estimated operating point is scaled or normalized to be compatible with the characteristics table. The table is entered to find the four neighboring ΔH, Q pairs. These are converted to prototype dimensions.

(3) A cubic interpolating function of ΔH v. Q is determined from the four points. Note that this function is applicable only in the vicinity of the estimated operating point.

(4) The slope of the cubic function is evaluated at the projected Q value. This is the term SLOPE in the energy equation (6.17).

(5) The value of ΔH at the projected Q is computed from the interpolating function. The CONST term on the energy equation (6.17) is then determined from SLOPE and the projected (ΔH, Q) point.

After solution of the entire system at each time step, the machine speed, ω, is evaluated in a separate calculation. The basic equation for speed change is

$$\frac{P}{\varpi} - T = I \frac{d\varpi}{dt} \tag{6.24}$$

where,

P	=	power supplied by the electric motor to the pump shaft
ω	=	rotational speed
I	=	moment of inertia of rotating parts
T	=	torque applied by the fluid on the impeller
$\frac{d\omega}{dt}$	=	angular acceleration

The actual finite difference form used in the program varies slightly with the different machine types. In order to evaluate this function, a second set of characteristics is required. These relate the variable ΔH or Q, ω, and, for turbines and pump-turbines, G, with a power, efficiency, or torque variable. With this table, the torque can be determined from the calculated machine operating point.

The computed discharge and speed are compared at the end of each time step with the estimated values. If these are not sufficiently close, the calculations for that time step are repeated. The new Q and ω estimates will be the values calculated in the previous iteration.

6.4.1 Pumps

Pump characteristics in WHAMO are expressed using dimensionless head difference, discharge, speed, and torque variables. These are ratios of actual values to the rated values for a particular machine. The first table is H/H_R versus Q/Q_R and $\tilde{\omega}/\tilde{\omega}_R$, and the second table is T/T_R versus Q/Q_R and $\tilde{\omega}/\tilde{\omega}_R$. The subscript R indicates a rated value. Figure 36 shows a graphical example of typical pump characteristics (see Knapp [1937]).

When a pump is operating it is restricted to synchronous speed and there is no speed change. When the pump is off, the speed change over the period t is calculated using

$$\Delta\varpi = \left(\frac{P}{\varpi} - T \right) \frac{\Delta t}{I} \tag{6.25}$$

The power, P, under these conditions will be zero.

6.4.2 Turbines

Characteristics used for turbines are based on test data for a model of specified size

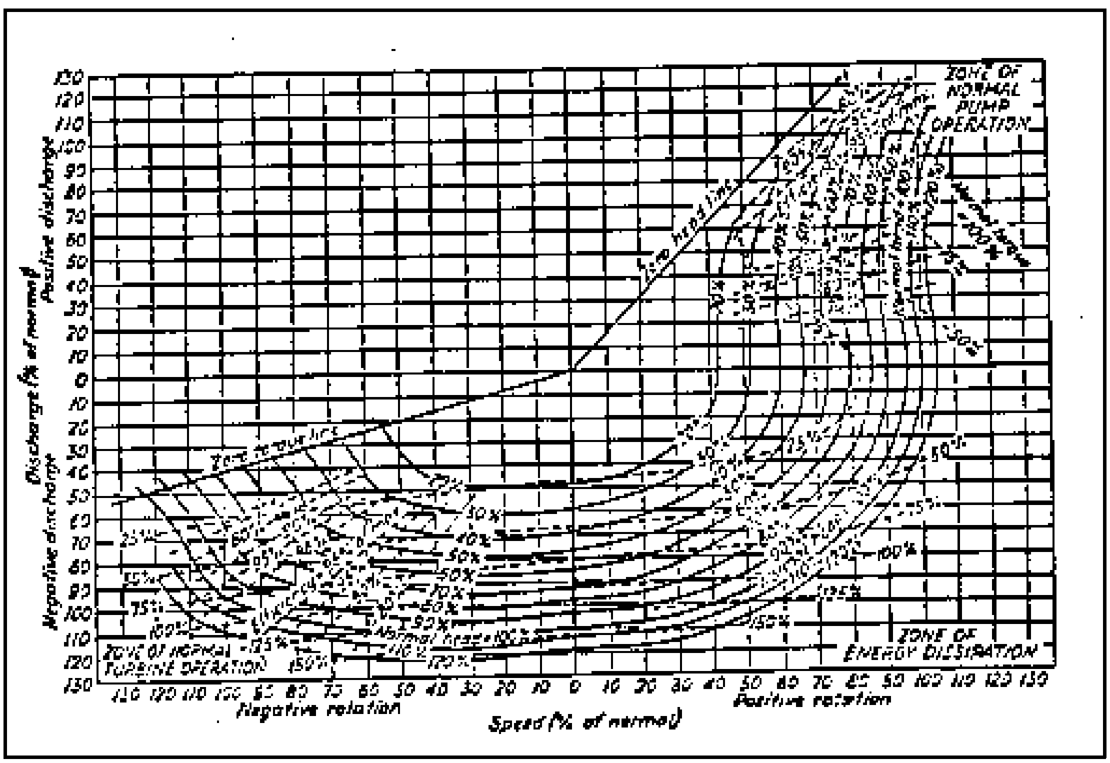


Figure 36. Complete characteristics diagram.

(usually 1 foot diameter) under 1 foot head. The tables are model discharge, Q_m, and model power, P_m, versus wicket gate positions and peripheral velocity factor, ϕ, where

$$\Phi = \frac{\frac{\omega D}{2}}{\sqrt{2g\Delta H}} \tag{6.26}$$

Other formulae for conversion between prototype data and model scale data are listed below, where the model head, ΔH_m, is 1 foot.

$$Q = Q_m \left(D/D_m \right)^2 \left(H/\Delta H_m \right)^{1/2} \tag{6.27}$$

$$P = P_m \left(D/D_m \right)^2 \left(H/\Delta H_m \right)^{3/2} \tag{6.28}$$

The user may choose to input characteristics in prototype scale. These are tables of Q and efficiency, η, versus H and at synchronous speed. The program converts these to model scale using the formulae above plus the formula for power given as

$$P = \eta \gamma Q \Delta H \cdot factor \tag{6.29}$$

Because Q is in the turbine characteristics table as a function of ϕ and G, the operating point cannot be directly located in the table from the G value and the estimated Q. Therefore, the bisection method is used to determine ϕ given G and Q. The method converges rapidly when applied to functions that are continuously monotonic (Perkins, 1964).

Note that ϕ is a function of both ω and ΔH. The user must take care that the range of ϕ values supplied in the characteristics tables will not be exceeded due to speed rise at load rejection or speed reduction with closed wicket gates. This applies as well to the implicit ϕ specified by ΔH at synchronous speed when prototype scale turbine characteristics are entered.

The new turbine speed at the end of time step is calculated using the following formula based on equation (6.24).

$$\omega^{n+1} = \sqrt{\omega^{n^2} + \Delta t \; (P^n + P^{n+1} - L^n - L^{n+1}) \; /I} \qquad \textbf{(6.30)}$$

The term L is the generator load and the superscripts indicate values at different time steps. When synchronous speed turbine operation is specified, the speed is not allowed to change and the generator load is constrained to be equal to the power delivered.

6.4.3 Pump-Turbine

As for turbines, pump-turbine characteristics are based on test data for a model of specified size under unit head. However, the organization of the data is different. Characteristics tables are stored in terms of a radial coordinate system, where operating locations are defined in terms of θ, the angle of a radial line drawn on a plot of discharge versus speed with the negative discharge axis, and r^2, the square of the radial distance from the origin (see Figure 37). There are two tables: one of r^2 versus θ and gate position and model torque, Tm, versus θ and gate position, where

θ = $\tan^{-1}$ (Nm/Qm)
r^2 = (Qm^2 + Nm^2)/Hm (radius squared)

and

Nm = model speed
Qm = model discharge
Hm = Model head = 1

This representation of characteristics is taken from Marchal (1965).

The operating point at the current time step is estimated in terms of Q, N, and G. Using the homologous relations

$$Q = QmD^2 \sqrt{\Delta H} \qquad \textbf{(6.31)}$$

$$N = Nm \sqrt{\Delta H}/D \qquad \textbf{(6.32)}$$

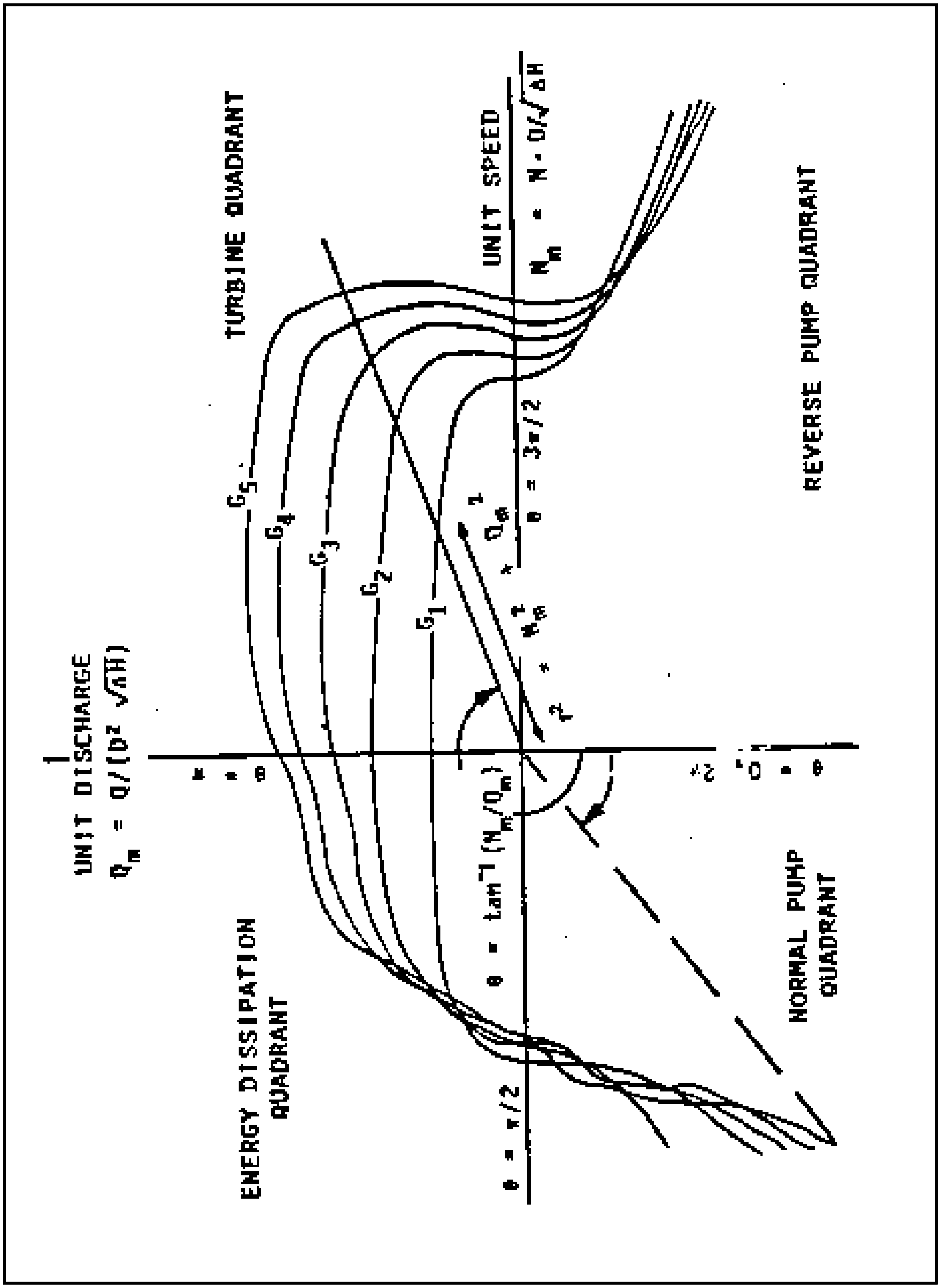


Figure 37. Speed-discharge characteristics plot for pump-turbines.

Θ can be determined from Q and N.

$$\theta_{est} = \tan^{-1} (D^3 N_{est} / Q_{est}), \quad Q_{est} \neq 0$$

$$\theta_{est} = 3\pi/2 \ , \quad Q_{est} = 0 \ and \ N > 0$$

$$\theta_{est} = \pi/2 \ , \quad Q_{est} = 0 \ and \ N < 0$$

The characteristics table can then be entered with θ_{est} and G to find r^2.

Neighboring table values of θ and r^2 can be converted using the homologous relations to ΔH and Q values from which a linear function of the form

$$\Delta H = SLOPE \cdot Q + CONST$$ (see Eq 6.17)

can be developed as for pumps and turbines.

Where N is near zero, another method of developing a ΔH-Q relation is adopted. Using the homologous relations, the expression for r^2 can be rewritten

$$r^2 = \frac{Q^2/D^4 + N^2 D^2}{\Delta H}$$

and then re-arranged to an expression of H as a function of Q

$$\Delta H = \left[\frac{1}{D^4 r^2} \right] Q^2 + \frac{N D^2}{r^2} \qquad (6.33)$$

the linear approximation about the estimated Q would be

$$\Delta H = SLOPE \cdot Q + CONST$$ (see Eq 6.17)

with

$$SLOPE = \frac{2 \ Q_{est}}{D^4 r^2}$$

$$CONST = \frac{Q_{est}^2}{D^4 r^2} + \frac{N_{est}^2 \ D^2}{r^2}$$

The speed change calculation uses equation 6.25 as for pumps. The torque table is entered with θ and gate to determine Tm, which is scaled up using the computed ΔH and the homologous relation

$$T = TmD^3\Delta H \tag{6.34}$$

6.5 Governor Algorithms

Mechanical governors respond to changes in speed while electrical governors typically respond to a combination of speed and power feedback. Both types are modeled in WHAMO with the following general differential equation.

$$a_1\frac{d^3G}{dt^3} + a_2\frac{d^2G}{dt^2} + a_3\frac{dG}{dt} + a_4(G-G_r) = a_5\frac{d^2\omega'}{dt^2} + a_6\frac{d\omega'}{dt} + a_7(\omega') + a_8 \tag{6.37}$$

where,

$\omega' = \omega - \omega_r$ (*mechanical governors*)

$\omega' = \omega_r - \omega - \theta(\frac{P}{P_r})$ (*electrical governors*

ω_r = *reference speed*

G_r = *reference gate position*

P = *generator output power*

P_r = *reference power*

and the coefficients a_i and θ are specified characteristics of the governing mechanism.

For a mechanical governor at initial steady state, equation 6.37 reduces to

$$a_4\ (G_o - G_r) = a_7\ (\omega_o - \omega_r) + a_8 \qquad \textbf{(6.37a)}$$

and therefore

$$G_r = G_o - \frac{a_7}{a_4}(\omega_o - \omega_r) - \frac{a_8}{a_4} \qquad \textbf{(6.37b)}$$

Substituting equation 6.37b into equation 6.37 and re-arranging terms results in the following expression written in terms of G_o and ω_o rather than G_r and ω_r.

$$a_1 \frac{d^3G}{dt^3} + a_2 \frac{d^2G}{dt^2} + a_3 \frac{dG}{dt} + a_4(G - G_O) \qquad \textbf{(6.37c)}$$

$$= a_5 \frac{d^2\omega^1}{dt^2} + a_6 \frac{d\omega^1}{dt} + a_7\ (\omega - \omega_o)$$

Following a similar procedure for electric governors, the general differential equation can be written

$$a_1 \frac{d^3G}{dt^3} + a_2 \frac{d^2G}{dt^2} + a_3 \frac{dG}{dt} + a_4(G - G_O)$$

$$= a_5 \frac{d^2\omega^1}{dt^2} + a_6 \frac{d\omega^1}{dt} + a_7\ (\omega_o - \omega \frac{\theta}{P_r}[P - P_o]) \qquad \textbf{(6.37d)}$$

Therefore, the following conventions are valid when applied to equation (6.37)

$$G_r = G_0$$

$$\omega^1 = \omega - \omega_0 \ (mechanical\ governors)$$

$$\omega^{/} = \omega_0 - \omega - \frac{\theta}{P_r}(P - P_0) \ (electrical\ governors)$$

$$a_8 = 0$$

Note that the assumption of constant coefficients, a_i, limits the applicability of the model to relatively small variations of gate and speed. The various limits and non-linearities of the governor and servo-mechanisms are not included.

The finite difference form of equation (6.37) used in the program code is as follows, where the bar $\overline{\ }$ indicates the known value of a variable from the previous step. A more complete derivation can be found in Perkins (1965).

$$G = b_2\bar{G} + b_3\bar{q} + b_4\bar{r} + b_5\bar{\omega}^1 + b_6\bar{s} + b_7 + b_8\omega^{/} \tag{6.38}$$

where,

$$q = \frac{2}{\Delta t}(G - \bar{G}) - \bar{q} \tag{6.39a}$$

$$r = \frac{4}{(\Delta t)^2}(G - \bar{G}) - \frac{4}{\Delta t}\bar{q} - \bar{r} \tag{6.39b}$$

$$s = \frac{2}{\Delta t}(\omega^1 - \bar{\omega}^1) - \bar{s} \tag{6.39c}$$

and,

$$b_1^{/} = 8a_1 + 4a_2(\Delta t) + 2a_3(\Delta t)^2 + a_4(\Delta t)^3 \tag{6.39d}$$

$$b_2' = 8a_1 + 4a_2(\Delta t) + 2a_3(\Delta t)^2 - a_4(\Delta t)^3 \quad \textbf{(6.39e)}$$

$$b_3' = 4(\Delta t)\ (2a_1 + a_2\Delta t) \quad \textbf{(6.39f)}$$

$$b_4' = 4a_1(\Delta t)^2 \quad \textbf{(6.39g)}$$

$$b_5' = -\Delta t[4a_5 + 2a_6(\Delta t) - a_7(\Delta t)^2] \quad \textbf{(6.39h)}$$

$$b_6' = -4a_5(\Delta t)^2 \quad \textbf{(6.39i)}$$

$$b_7' = 2(\Delta t)^3[a_4G_r + a_8] \quad \textbf{(6.39j)}$$

$$b_8' = \Delta t[4a_5 + 2a_6(\Delta t) + a_7(\Delta t)^2] \quad \textbf{(6.39k)}$$

and,

$$b_i = \frac{b_i'}{b_1'} \quad (i = 1,2,...8) \quad \textbf{(6.39l)}$$

The above represents a simple (though messy) relation between the two unknowns, G and ω', at any time step. In the turbine or pump-turbine simulation, under governor control, the wicket gate position cannot be determined from a specified schedule. Instead, the governor equation above is solved for G using the projected value of ω'. Recall that if the projected ω' does not agree well with the value calculated after solution of the entire system, the calculation for that time step is repeated.

An additional WHAMO governor option allows the specification of maximum gate opening and closing rates. When this option is in effect, neither the governor equation or its coefficients are altered. Simply, when the new gate position is calculated the implicit rate is compared with the specified maximum. If the rate is excessive the new gate position is adjusted so that the rate of gate movement is equal to the maximum rate allowed.

6.5.1 Mechanical Governors

Figure 38 shows the block diagram for a typical speed-droop type mechanical governor. The term S is the Laplace transform operator and the other parameters shown are defined for equation (6.40) below. The differential equation described by the block diagram is written below in terms of normalized, or "per unit", G and ω values.

$$\frac{d^2}{dt^2}\left[\frac{G}{100\%}\right] + \left[\frac{T_G+(\delta+\sigma)T_r}{T_G T_r}\right]\frac{d}{dt}\left[\frac{G}{100\%}\right] \qquad (6.40)$$

$$+ \left[\frac{\sigma}{T_G T_r}\right]\left[\frac{(G-G_r)}{100\%}\right]$$

$$= \left[-\frac{1}{T_G}\right]\frac{d}{dt}\left[\frac{\omega'}{\omega_0}\right] + \left[-\frac{1}{T_G T_r}\right]\left[\frac{\omega'}{\omega_0}\right]$$

where, T_G = characteristic time of promptitude of the governor; i.e., the proportionality constant relating speed error to wicket gate velocity

T_r = dashpot relaxation time constant

σ = permanent speed droop

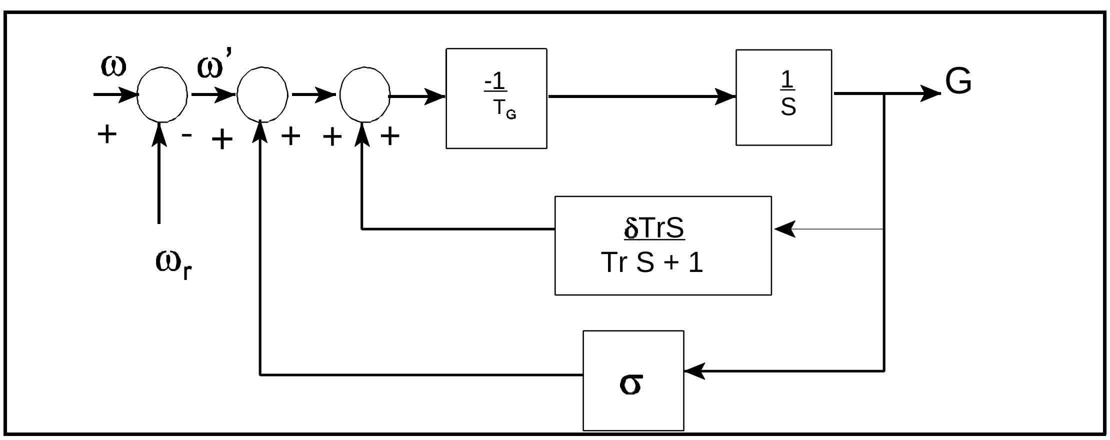


Figure 38. Block diagram for mechanical governor.

δ = temporary speed droop

With reference to equations (6.40) and (6.37), the following relations can be written:

$$a_1 = 0.0 \quad \text{(6.41a)}$$

$$a_2 = 1.0 \quad \text{(6.41b)}$$

$$a_3 = \frac{T_G + T_r(\sigma + \delta)}{T_G T_r} \quad \text{(6.41c)}$$

$$a_4 = \frac{\sigma}{T_G T_r} \quad \text{(6.41d)}$$

$$a_5 = 0.0 \quad \text{(6.41e)}$$

$$a_6 = -\frac{100}{\omega_0 T_G} \quad \text{(6.41f)}$$

$$a_7 = -\frac{100}{\omega_0 T_G T_r} \quad \text{(6.41g)}$$

$$a_8 = 0.0 \quad \text{(6.41h)}$$

WHAMO uses these relations to compute the coefficients a_i from the mechanical governor parameters. The following are the default values for the mechanical governor coefficients. These are based on COE test results and Hagihara (1979).

$$T_G = 0.15 \ \textit{seconds}$$

$$T_r = 3.33 \ T_w \ \textit{seconds}$$

$$T_w = \frac{\Sigma LV}{g\Delta H}, \ \textit{water starting time}$$

$$\Sigma LV = l_1 v_1 + l_2 v_2 + \ldots + l_n v_n$$

l = length of conduit of constant area
v = associated velocity at rated discharge (fps)
ΔH = rated net head (feet)

$$\sigma = 0.05$$

$$\delta = 1.25 \; T_w / T_m$$

$$T_m = \frac{wR^2 n_0^2}{1{,}610{,}000P}, \; \textit{mechanical starting time} \tag{6.43}$$

$$WR^2 = \textit{total inertia}(lb - ft^2)$$

WR^2 = total inertia (lb-ft^2)
n_0 = synchronous speed (rpm)
P = rated output (HP)

6.5.2 Electrical Governors

The block diagram for a proportional-integral-differential (PID) electrical governor with power generation feedback is shown in Figure 39. The differential equation for this, with G and ω^1 again expressed per unit, is

$$0.0225 \; T_t \frac{d^4}{dt^4} \left(\frac{G}{100\%}\right) + (0.0225 + 0.30T_t) \frac{d^3}{dt^3} \left(\frac{G}{100\%}\right) \tag{6.44}$$

$$+ (T_t + 0.30) \frac{d^2}{dt^2} \left(\frac{G}{100\%}\right) + \frac{d}{dt} \left(\frac{G}{100\%}\right)$$

$$= K_d \frac{d^2}{dt^2} \left(\frac{\omega'}{\omega_0}\right) + K_p \frac{d}{dt} \left(\frac{\omega'}{\omega_0}\right) + K_i \left(\frac{\omega'}{\omega_0}\right)$$

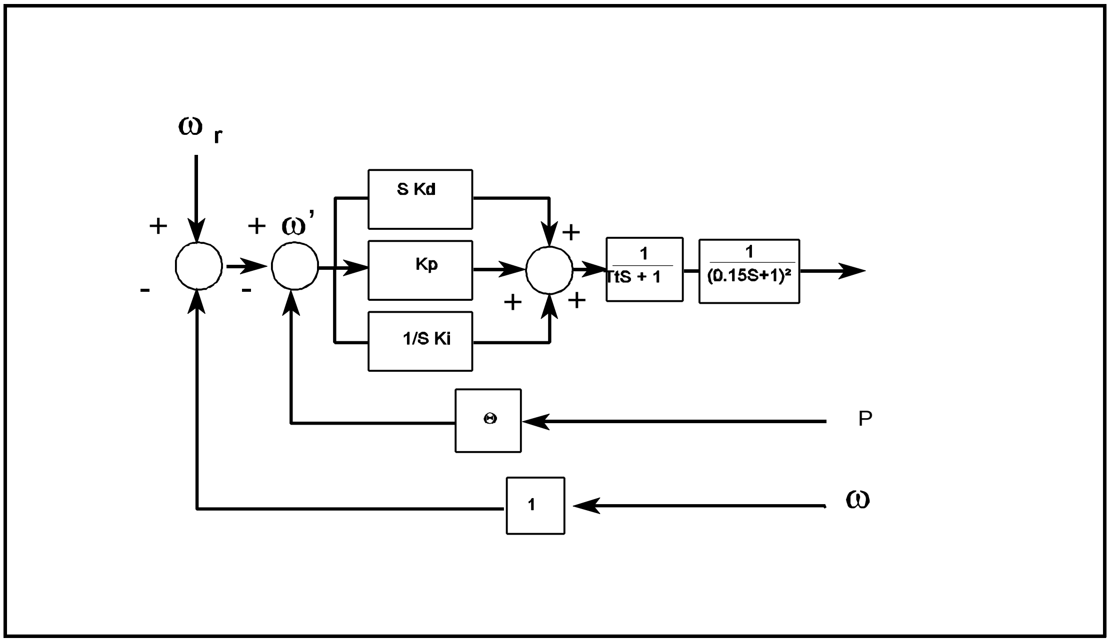


Figure 39. Block diagram for electrical governor.

in which T_t = pilot servo-motor time constant
K_d = derivative gain
K_p = proportional gain
K_i = integral gain
θ = speed regulation term (similar to droop)

Since T_t is of the order of 0.05, the coefficient of the fourth order term is about 1/1000 times the coefficient of the first order term. The fourth order term is therefore neglected.

The relations between the coefficients of the general governor equation (6.37) and the electrical governor input parameters are as follows:

$$a_1 = 0.0225 + 0.30\,T_t \quad \text{(6.45a)}$$

$$a_2 = T_t + 0.30 \quad \text{(6.45b)}$$

$$a_3 = 1.0 \quad \text{(6.45c)}$$

$$a_4 = 0.0 \tag{6.45d}$$

$$a_5 = \left(\frac{100}{\omega_0}\right) K_d \tag{6.45e}$$

$$a_6 = \left(\frac{100}{\omega_0}\right) K_p \tag{6.45f}$$

$$a_7 = \left(\frac{100}{\omega_0}\right) K_i \tag{6.45g}$$

$$a_8 = 0.0 \tag{6.45h}$$

COE-recommended parameter values computed by default are given below:

$$K_i = 0.24 \frac{T_m}{T_w^2}$$

$$K_p = 0.8 \frac{T_m}{T_w}$$

$$K_d = 0.27 T_m$$

$$T_t = 0.05$$

$$\theta = 0.05$$

The normal range of adjustment for the gains is:

	Online	Offline
K_p	0.5 - 18	0.14 - 4.6
K_i	0.3 - 7	0.1 - 1.8
K_d	0.0 - 1.8	0.0 - 1.8

6.6 Algorithms for Surge Tanks

6.6.1 Simple Surge Tanks

The simple surge tank is modeled as a single segment conduit with a variable free surface boundary. The conduit equations of Section 6.2 are used. For a surge tank the length of the segment varies as the water surface rises and falls. The segment length used in the computation is calculated from the water surface elevation at the previous time step. Since the surge tank area may vary with elevation, the average segment area used in the conduit computations is calculated by dividing the total water volume in the tank at the previous time step by the segment length.

The governing equation at the free surface boundary is:

$$\frac{dH}{dt} - \frac{Q}{A}\left[\frac{1}{gA}\frac{dQ}{dt} + 1\right] = 0 \qquad \textbf{(6.46)}$$

where A is the surge tank area at the water surface.

The finite difference form of this equation is:

$$H = TN - TA \cdot Q \qquad \textbf{(6.47)}$$

$$\textit{where,} \qquad TA = -\frac{\Delta t}{2A}\left[\frac{1}{gA}\left[\frac{Q^{n-1} - Q^{n-2}}{\Delta t}\right] + 1\right] \qquad \textbf{(6.48a)}$$

$$TN = H^{n-1} - TA \cdot Q^{n-1} \qquad \textbf{(6.48b)}$$

Refer to Resource Analysis (1978) and Resource Analysis (1976) for further treatment.

6.6.2 Surge Tank Outflow

Outflow from a surge tank may occur when the tank is overtopped or when the water surface reaches the level of a user specified outflow port. Outflow from a surge tank is considered lost to the system.

Flow through a surge tank outlet is accounted for "below" the water surface. That is, prior to solving the free surface boundary equation (6.46), the outflow rate is subtracted from the water surface discharge.

$$Q_j = Q_{j-1} - OF \tag{6.49}$$

where OF is the outflow rate and location j is at the water surface and j-1 is below the outlet.

The outflow rate can be expressed as a function of head using orifice or weir equations. For a rectangular orifice, the equation is:

$$OF = C\ 2/3\ \sqrt{2g}\ B \left[(H-ELOUT)^{3/2} - (H-ELOUT-HOUT)^{3/2} \right] \tag{6.50}$$

in which
- B = width of outlet
- C = discharge coefficient
- ELOUT = bottom elevation of outlet
- HOUT = height of outlet

This equation also applies for weir flow when the water surface elevation is less than the top of the orifice. By invoking the rule

$$if\ (H-ELOUT-HOUT) \le 0, \quad set\ (H-ELOUT-HOUT) = 0$$

a discharge coefficient of 0.61 is equivalent to a weir coefficient of 3.26. Equation (6.50) can be approximated linearly in the form:

$$OF = mH + b \tag{6.51}$$

with

$$m = C\ \sqrt{2g}\ B \left[(H^{n-1}-ELOUT)^{1/2} - (H^{n-1}-ELOUT)^{1/2} \right] \tag{6.52a}$$

and

$$b = -mH^{n-1} + C\ 2/3\ \sqrt{2g}\ B \left[(H^{n-1}-ELOUT)^{3/2} - (H^{n-1}-ELOUT-HOUT)^{3/2} \right] \tag{6.52b}$$

Three relations are given for circular orifice outflow for different depth to diameter ratios. These are:

$$1)\ OF = 0.48\ D^2\ \sqrt{gD} \left[\frac{H-ELOUT}{D} \right]^{1.9},\ 0 < \frac{H-ELOUT}{D} < 0.8 \tag{6.53}$$

$$2)\ OF = 0.44\ D^2 \sqrt{gD} \left[\frac{H-ELOUT}{D}\right]^{1.5},\ 0.8 < \frac{H-ELOUT}{D} < 1.2 \qquad \textbf{(6.54)}$$

$$3)\ OF = C \cdot \frac{\pi}{4} D^2 \sqrt{2g}\ (H-ELOUT-D/2)^{0.5},\ \frac{H-ELOUT}{D} > 1.2 \qquad \textbf{(6.55)}$$

These three equations are linearized in the form of equation (6.51) with:

CASE 1

$$m = 1.9 \left[\frac{0.48 D^2 \sqrt{gD}}{D^{1.9}}\right] (H^{n-1} - ELOUT)^{0.9} \qquad \textbf{(6.56a)}$$

$$b = -mH^{n-1} + \frac{1}{1.9}\ m\ (H^{n-1} - ELOUT) \qquad \textbf{(6.56b)}$$

CASE 2

$$m = 1.5 \left[\frac{0.44\ D^2 \sqrt{gD}}{D^{1.5}}\right] (H^{n-1} - ELOUT)^{0.5} \qquad \textbf{(6.57a)}$$

$$b = -mH^{n-1} + \frac{1}{1.5}\ m\ (H^{n-1} - ELOUT) \qquad \textbf{(6.57b)}$$

CASE 3

$$m = 0.5\ \left(C\ \frac{\pi}{4} D^2 \sqrt{2g} \right)\ (H^{n-1} - ELOUT - D/2)^{-0.5} \qquad \textbf{(6.58a)}$$

$$b = -mH^{n-1} + \frac{1}{0.5}\ m\ (H^{n-1} - ELOUT - D/2) \qquad \textbf{(6.59b)}$$

Surge tank overtopping is accounted for at the water surface by modifying the free surface boundary equation (6.46) to the form

$$\frac{dH}{dt} = \frac{1}{A}\left[\frac{1}{gA}\ \frac{dQ}{dt} + 1\right](Q - OT) \qquad \textbf{(6.60)}$$

where OT is overtopping rate. The overtopping rate can be computed using the rectangular orifice equation (6.52) with

$$B = \sqrt{4\pi A}\ ,\ circumference$$

and

$$(\ H - ELOUT - HOUT\ ) = 0$$

Replacing OT in equation (6.60) with mH + b, the finite difference form of the surge tank boundary equation with outflow is

$$H = TN - TA \cdot Q \qquad \textit{(see Eq 6.47)}$$

$$TA = -\frac{1}{2}\ TERM\ /\ (1 + m{\cdot}TERM) \qquad \textbf{(6.60a)}$$

$$TN = -TA{\cdot}Q^{n-1} - TERM{\cdot}b\ /\ (1-m{\cdot}TERM) + H^{n-1}/(1 + m{\cdot}TERM)$$

$$TERM = \frac{\Delta t}{A}\left[\frac{1}{gA}\ \frac{(Q^{n-1} - Q^{n-2})}{\Delta t} + 1\right] \qquad \textbf{(6.60b)}$$

6.6.3 Differential Surge Tanks

A definition sketch of the model representation of a differential surge tank is shown in Figure 40. The computation is carried out for the riser in a manner quite similar to a simple surge tank. The main difference is that flows into and out of the riser at the top or at the orifice connection to the outer tank must be accounted for. The water level in the outer tank is accounted separately, rather than solved simultaneously with the rest of the system. This is reasonable because transients will be much more gradual there than in the rest of the system. Also, because velocities will be very slow (≤ 1 fps) friction and water hammer effects in the outer tank will be ignored.

The simulation procedure at each time step involves first making a projection of HTANK, the water level in the outer tank, based on values at the previous time steps. The estimated HTANK is used to set up the riser equations for the first and last segments (see below) and the free surface boundary equation. After solution of the entire system HTANK is calculated from the hydraulics of the surge tank.

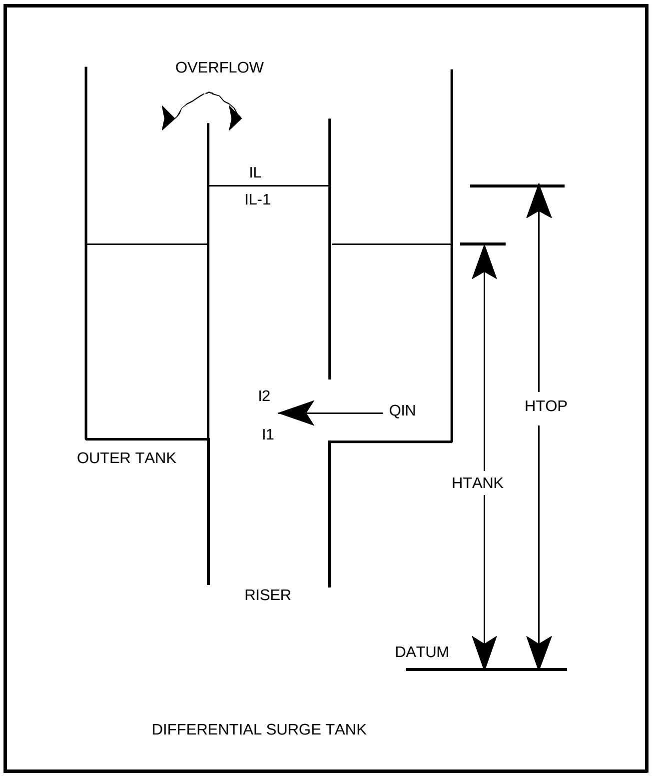


Figure 40. Differential surge tank.

If HTANK is above the top of the riser, the solution may be influenced by a small error in the projected HTANK, in which case the calculated value is compared with the projection and, if agreement is unsatisfactory, the entire system computation is reiterated for that time step.

The equations governing the riser modeling and the water level calculations in the outer tank are presented on the following page. The equations for the first segment from internal node I1 to I2 are the following:

Energy: $$H_{I2} = H_{I1} \tag{6.61}$$

Continuity: $$Q_{I2} = Q_{I1} + QIN \tag{6.62}$$

where $$QIN = \pm C_d \cdot A_D \sqrt{2g} \sqrt{|HTANK - H_{I1}|} \tag{6.63}$$

C_d = *discharge coefficient of the orifice*

A_D = *area of the orifice*

The plus sign applies where HTANK≥ H_{I1} and the minus sign where HTANK < H_{I1}. Equation (6.62) is linearized to the form

$$Q_{I2} = Q_{I1} + SLOPE \cdot H_{I1} + CONST \tag{6.64}$$

where $$SLOPE = -1/2\ (C_d\ A_O\ \sqrt{2g})\ (|H_{I1}^{n-1} - HTANK^{n-1}|)^{-1/2} \quad \textbf{(6.65a)}$$

$$CONST = -\ SLOPE \cdot H_{I1}^{n-1} + (Q_{I2}^{n-1} - Q_{I1}^{n-1}) \quad \textbf{(6.65b)}$$

The equations for the intermediate segment(s) from I2 to IL-1 are simply the continuity and momentum equations of Section 6.2 as applied to a simple surge tank.

The last segment, from IL-1 to IL, is added to account for possible overflow from the outer tank into the riser.

Energy: $$H_{IL} = H_{IL-1} \quad \textit{(See Eq 6.61)}$$

Continuity: $$Q_{IL} = Q_{IL-1} + QADD \quad \textbf{(6.66)}$$

QADD is overflow from the outer tank into the riser.

$$QADD = O : H_{IL} > HTOP \quad \textbf{(6.67a)}$$

$$QADD = O : HTANK < HTOP \quad \textbf{(6.67b)}$$

$$QADD = C_w \cdot CIRC(HTANK - HTOP)^{3/2} : HTANK > HTOP \text{ and } H_{IL} \leq HTOP \quad \textbf{(6.67c)}$$

The term C_w is a weir coefficient and CIRC is the circumference of the riser.

The boundary equation at the last node of a differential surge tank is essentially the same as that shown previously for a simple surge tank, expanded to account for overflow from the riser into the main tank. Additionally, the area term used in the calculation can be that of the riser or the riser plus outer tank depending on the hydraulic conditions.

Boundary Equation

$$H_{IL} = TN - TA \cdot Q_{IL} \qquad \textit{(see Eq 6.47)}$$

$$TA = -1/2\ TERM\ /\ (1 + m \cdot TERM) \tag{6.68a}$$

$$TN = -TA \cdot Q^{n-1} - TERM \cdot b\ /(1 + 1m \cdot TERM) + H^{n-1}/(1 + m \cdot TERM) \tag{6.68b}$$

$$TERM = \frac{\Delta t}{A} \left[\frac{1}{gA} \frac{(Q^{n-1} - Q^{n-2})}{\Delta t} + 1 \right] \tag{6.68c}$$

$$m = \frac{3}{2} \cdot C_w \cdot CIRC \cdot (H_{IL}^{n-1} - HTOP)^{1/2} \tag{6.68d}$$

$$B = -m \cdot H^{n-1} + C_w \cdot CIRC \cdot (H_{IL}^{n-1} - HTOP)^{3/2} \tag{6.68e}$$

If $H_{IL} < HTOP$

m = 0

b = 0

A = area of riser

If $H_{IL} > HTOP$ and $HTANK > HTOP$

m = 0

b = 0

A = area of outer tank + riser

The above finite difference equation was developed using the methods of Section 6.6.2.

Prior to the transient claculation at each time step, the head in the main tank is estimated from the previous values using:

$$HTANK^{n} = HTANK^{n-1} + (HTANK^{n-1} - HTANK^{n-2}) / 2 \tag{6.69}$$

This should provide a reasonable estimate as HTANK will change slowly and "smoothly" compared to other system heads and discharges.

At the end of each time step, HTANK is computed from the hydraulic conditions as follows:

If HTANK < HTOP and H_{IL} < HTOP (no overflow):

$$HTANK^{n} = HTANK^{n-1} + \frac{DT}{2AT}(QIN^{n} + QIN^{n-1}) \tag{6.70a}$$

$$QIN = Q_{I1} - Q_{I2}$$

$$AT = area\ of\ outer\ tank$$

If HTANK > HTOP and H_{IL} > HTOP:

$$HTANK^{n} = H(NL) \tag{6.70b}$$

If $HTANK^{est}$ > HTOP and H(NL) < HTOP:

$$HTANK^{n} = HTANK^{n-1} + \frac{DT}{2AT}(QIN^{n} + QIN^{n-1})$$

$$- \frac{DT}{AT} \cdot C_{W} \cdot B\ ([HTANK^{n-1} + HTANK^{est}] / 2.0)^{3/2} \tag{6.70c}$$

If $HTANK^{est}$ < HTOP and H(NL) > HTOP:

$$HTANK^{n} = HTANK^{n-1} + \frac{DT}{2AT}(QIN^{n} + QIN^{n-1})$$

$$+ \frac{DT}{AT} \cdot C_{w} \cdot ([H^{n}(NL) + H^{n-1}(NL)]/2.0)^{3/2}$$

6.6.4 Air Chamber Surge Tanks

The pressure-volume relationship of the air in the tank is incorporated in the air chamber surge tank boundary equation. The basis of the calculations is the ideal gas law

$$pV = mRT \tag{6.71}$$

with p = absolute pressure
V = air volume
m = air mass
R = gas constant for air
T = absolute temperature

The algorithm presented includes the possibility of air release or air intake to the tank.

Air chambers and hydropneumatic surge tanks are most commonly modeled using the polytropic equation, which is an approximation of equation (6.71).

$$pV^n = C \tag{6.72}$$

The value of the polytropic exponent, n, may vary from 1.0 for isothermal processes to 1.4 for adiabatic processes. C is normally a constant depending on the initial state of the air. However, C may change as a result of air entry or release.

To develop a differential equation relating head and discharge at the water surface boundary we start by differentiating the polytropic equation (6.72) with respect to time.

$$\frac{dp}{dt} = V^{-n}\frac{dC}{dt} + C(-n)V^{-n-1}\frac{dV}{dt}$$

The rate of air volume change is the negative rate of water discharge:

$$\frac{dV}{dt} = -Q \tag{6.74}$$

C varies only if the air mass, m, varies due to venting. Combining equations (6.71) and (6.72)

$$C = mRTV^{n-1} \tag{6.75}$$

$$\text{and} \qquad \frac{dC}{dm} = RTV^{n-1} \tag{6.76}$$

If the rate of air inflow, dm/dt, is known, then

$$\frac{dC}{dt} = \frac{dC}{dm}\frac{dm}{dt} = RTV^{n-1}\frac{dm}{dt} \tag{6.77}$$

Substituting equations (6.74) and (6.77) into (6.73), gives

$$\frac{dp}{dt} = \frac{RT}{V}\frac{dm}{dt} + CnV^{-n-1}Q \tag{6.78}$$

Recognizing that

$$\frac{RT}{V} = \frac{p}{m}$$

and

$$C = pV^{n}$$

equation (6.78) can be written

$$\frac{dp}{dt} = \frac{p}{m}\frac{dm}{dt} + \frac{pn}{V}Q \tag{6.79}$$

Next, the absolute pressure, p, is converted to total head, H, using the equation

$$P = (H - \frac{Q^2}{2gA^2} - WS + HB)\,\rho g \tag{6.80}$$

where WS = water surface elevation
HB = atomospheric pressure
ρ = mass density of water

Recognizing that

$$\frac{dWS}{dt} = \frac{Q}{A} \tag{6.81}$$

the derivative can be expressed

$$\frac{dp}{dt} = \left[\frac{dH}{dt} - \frac{Q}{gA^2}\frac{dQ}{dt} - \frac{Q}{A} \right] \rho g$$

Substituting the above expressions for p and $\frac{dp}{dt}$ into equation (6.79) results in the following boundary equation for a vented air chamber:

$$\frac{dH}{dt} = \frac{1}{m}\frac{dm}{dt}H + \frac{n}{V}QH + \left[\frac{1}{gA^2}\frac{dQ}{dt} + \frac{1}{A} - \frac{n}{V}\left(\frac{Q^2}{2gA^2} + WS - HB\right)\right]Q$$

$$- \frac{1}{m}\left[\frac{Q^2}{2gA^2} + WS - HB \right]\frac{dm}{dt} \tag{6.83}$$

To convert equation (6.83) to linear, finite difference form, the following approximations are made:

1) An estimated value of Q is used to compute V and $(\frac{n}{V}Q)$. Minor inaccuracies should be insignficant given that QΔt << V.

2) An estimated Q is used to compute velocity head related terms. These are minor terms.

3) $\frac{dm}{dt}$ is assumed known for the time step. Note that for small time steps $\frac{1}{m}\frac{dm}{dt}\Delta t \ll 1$. Formulae for computing $\frac{dm}{dt}$ will be presented later in the subsection.

The resulting expression is written below.

$$H = TN - TA \cdot Q \qquad \text{(see Eq 6.47)}$$

$$TA = \frac{\left(\frac{1}{gA^2}\frac{d\bar{Q}}{dt} + \frac{1}{A} - \frac{TERM1 \cdot n}{V}\right)\Delta t}{2 - TERM2} \tag{6.84a}$$

$$TN = -TA \cdot Q^{n-1} - \frac{2 \cdot TERM1}{2 - TERM2}\frac{1}{m}\frac{dm}{dt}\Delta t + \frac{2 + TERM2}{2 - TERM2}H^{n-1} \tag{6.84b}$$

$$TERM1 = \frac{\bar{Q}^2}{2gA^2} + WS - HB \tag{6.84c}$$

$$TERM2 = \left(\frac{1}{m}\frac{dm}{dt} - \frac{n}{V}\bar{Q}\right)\Delta t \tag{6.84d}$$

Note that $\bar{Q}, \frac{d\bar{Q}}{dt}, A, V, WS, m,$ and $\frac{dm}{dt}$ are estimated average values for the time step.

The air mass flow rate through an orifice is computed using formulae given by Wylie (1978).

$$\frac{dm}{dt} = \pm CA\ p_1 \sqrt{\frac{7}{RT_1}\left[\left(\frac{p_2}{p_1}\right)^{1.4286} - \left(\frac{p_2}{p_1}\right)^{1.714}\right]},\ p_2 > 0.53\ p_1 \tag{6.85}$$

$$\frac{dm}{dt} = \pm\ C \cdot A\ \frac{0.686}{\sqrt{RT_1}}\ p_1,\ p_2 \le 0.53\ p_1 \tag{6.86}$$

p_1 = the greater of tank pressure and atmospheric pressure

p_2 = the lesser of tank pressure and atmospheric pressure

C = orifice discharge coefficient

A = orifice area

T_1 = temperature associated with p_1

C•A as a function of tank pressure is supplied by the user, as well as atmospheric pressure and temperature. Tank pressure and temperature computed at the previous time step are used. If atmospheric pressure exceeds tank pressure, $\frac{dm}{dt} \geq 0$.

An alternative method of modeling air chambers, known as the rational heat transfer (RHT) method, has been developed by Graze (1968) and Graze, Schubert, and Forrest (1976). This method solves the ideal gas equation (6.71) without making the assumptions implicit in the empirical polytropic equation (6.72). Rather, all energy transfers are accounted for explicitly.

Following a similar procedure to that shown for the polytropic process, the following vented air chamber boundary equation can be developed:

$$\frac{dH}{dt} = \frac{1}{m}\frac{dm}{dt}H + \frac{1.4}{V}Q\ H + \left[\frac{1}{gA^2}\frac{dQ}{dt} + \frac{1}{A} - \frac{1.4}{V}(\frac{Q^2}{2gA^2} + WS\right.$$

$$-\frac{1}{m}\left[\frac{Q^2}{2gA^2} + WS - HB\right]\frac{dm}{dt} + \frac{0.4}{V}\frac{1}{2g\rho g}v_{air}^2\frac{dm}{dt} - \frac{0.4}{V}\frac{1}{\rho g}\frac{dZ}{dt} \quad \textbf{(6.87)}$$

v_{air} = air velocity passing orifice

$\frac{dZ}{dt}$ = heat exchange rate between tank and surroundings, not including air flow

The RHT boundary equation (6.87) is the same as the polytropic boundary equation (6.83) with no energy transfer (n = 1.4), plus two explicit terms to account for energy transfer. The first term, which accounts for the flow energy of air leaving the system, is small enough to be neglected. The heat transfer rate is calculated using the formula suggested by Graze (1968).

$$\text{where} \quad \frac{dZ}{dt} = 0.19 \; | \; T_{in} - T_{ex} \; |^{1/3} \; (T_{in} - T_{ex}) \; SA \tag{6.88}$$

T_{in} = air temperature in tank
T_{ex} = surrounding temperature
SA = tank surface area for heat exchange

Note that latent heat effects, which could be significant if the air chamber temperature passes through freezing or boiling temperature, are not considered.

The user has the option of specifying either the polytropic or the RHT method for air chamber calculations.

6.7 Algorithms for Junctions

6.7.1 Simple Junctions

There is no head loss across a simple junction. The governing equations are these:

$$\textit{Continuity}: \quad Q_j + Q_{j+1} + Q_{j+2} \; .... = 0 \tag{6.89}$$

$$\textit{Energy}: \quad H_j = H_{j+1} = H_{j+2} \; .... \tag{6.90}$$

In programming continuity at junctions, care must be taken that the correct sign is used with Q. The convention used by the program for continuity may differ from the convention assigned by the user to a particular branch. The energy relations shown above are not included explicitly in the matrix of system equations. Rather, the coefficients of H, H_{j+1}, ..., for the respective serial branch outer equations are all placed in the same column of the system matrix, thus equating them implicitly.

6.7.2 Tee Junctions

The algorithms for Tee Junctions differ from simple junctions in that Tee Junctions are restricted to three linking branches (of certain geometry) and head losses are calculated from branch to branch. The continuity equation is the same as for a simple junction

$$Q_j + Q_{j+1} + Q_{j+2} = 0$$ *(see Eq 6.89)*

and there are two energy equations of the form:

$$H_j - H_{j+2} = HL_{j \cdot j+2}$$

$$H_{j+1} - H_{j+2} = HL_{j+1 \cdot j+2}$$ *(see Eq 6.21)*

A complete description of the head loss, HL, calculation can be found in Resource Analysis (1978). This procedure was originally presented in Gardel (1965). The Gardel coefficients can be modified to achieve a specified head loss coefficient from the upstream conduit to the riser when there is no flow in the riser. Such a modification affects head loss calculations for other flow regimes as well.

6.8 Special Options for Valve, Gate, and B.C. Schedules

6.8.1 Superimposed Sinusoidal Variation

To superimpose a sinusoidal variation on a valve, gate, load, reservoir elevation, or flow boundary schedule, the following equation is applied:

$$BC_{\mathrm{mod}}(t) = BC_{sched}(t) + sine[2\pi(t \cdot FREQ + PHASE/360)] \cdot AMP \qquad \textbf{(6.91)}$$

BC_{mod}	=	modified boundary condition value
Bc_{sched}	=	boundary condition value from schedule
FREQ	=	specified sinusoidal frequency
PHASE	=	specified phase angle
AMP	=	specified amplitude

6.8.2 Gate Closure

The standard gate closure consists of four distinct portions: gate acceleration, linear closure, gate deceleration, and final closure at a reduced constant rate.

Alternatively, a non-linear cushioning curve may be specified in place of gate deceleration and linear final closure.

The acceleration portion is defined by the user as the time to reach the straight line closure rate, TACC. Assuming a constant rate of acceleration,

$$at \qquad t_1 = t_0 + TACC,$$

$$G(t_1) = G(t_0) - \frac{1}{2}\, TACC\, \frac{100}{RTCLOS} \tag{6.92}$$

where

RTCLOS = user defined closure time for full stroke at constant velocity

The interpolation routine will provide a cubic curve between the points (t_0 , G [t_0]) and (t_1, G [t_1]).

From the gate value G(t_1), at which full velocity has been reached, to GCUSH, where the cushioned closure begins, gate position is determined by

$$G(t) = G(t_1) - (t - t_1)\, \frac{100}{RTCLOS} \tag{6.93}$$

A constant closure rate applies until time t_2 which is calculated

$$t_2 = t_1 + (\, G[t_1] - GCUSH\,)\, \frac{RTCLOS}{100} \tag{6.94}$$

The gate deceleration time, TDEC, is the time specified by the user for the gate closure rate to be reduced to the final cushioned rate. The computations are similar to those for gate acceleration.

$$at \qquad t_3 = t_2 + TDEC$$

$$G(t_3) = G(t_2) - \frac{1}{2}\ TDEC\ \frac{20}{RTCLOS + RTCUSH} \qquad \textbf{(6.95)}$$

where

RTCUSH = user defined closure rate in terms of time for full stroke

The final closure at constant rate is computed

$$G(t) = G(t_3) - (t - t_3)\ \frac{100}{RTCUSH} \qquad \textbf{(6.96)}$$

The gate will be closed at time t_4 and remain closed thereafter.

$$t_4 = t_3 + G(t_3)\ \frac{RTCUSH}{100} \qquad \textbf{(6.97)}$$

A non-linear final closure curve (Figure 41) may be defined by a set of G/GCUSH versus $(t-t_2)$/TCUSH points, where TCUSH is the specified time from the beginning of cushioned closure to zero gate. Each point ofthe dimensionless curve, is converted to a point in the gate schedule using the specified GCUSH and TCUSH values and the computed t_2.

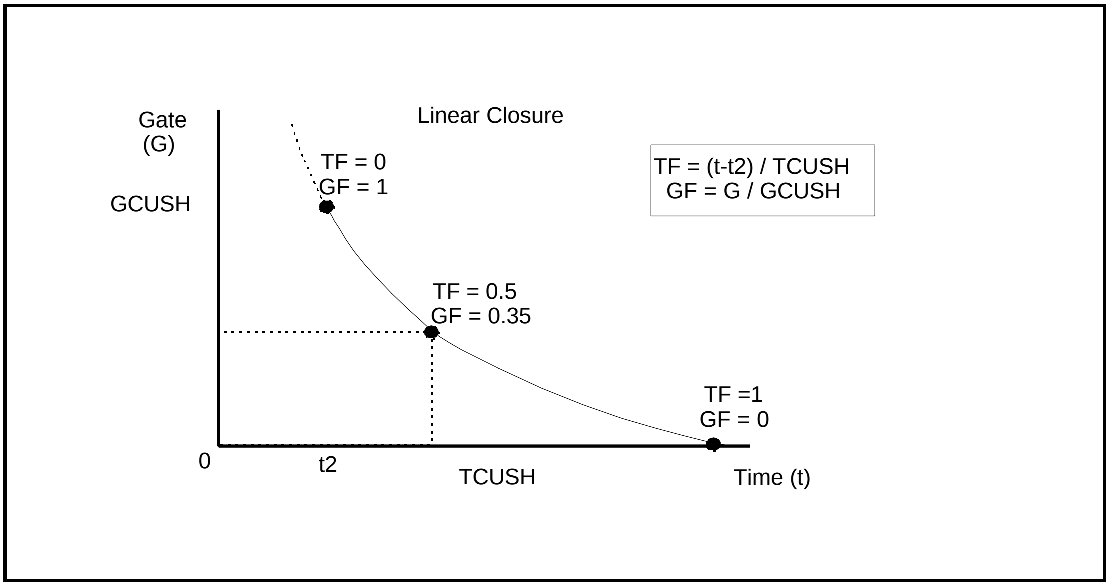


Figure 41. Non-linear final closure curve.

6.9 Branch Reduction Scheme

The system inner equations of Sections 6.1 - 6.8 relate the heads and discharges at adjacent internal nodes using continuity and momentum (or energy) conservation principles as applied to the element type which contains the two nodes. As discussed in Section 2.6 the inner equations of each serial branch are reduced to a single pair of branch outer equations relating, respectively, the head at the downstream node and the discharge at the downstream node to the head and discharge at the upstream node. These are then solved simultaneously with the junction branch equations.

This process, termed forward construction, is quite similar to the reduction process described for conduits by Resource Analysis, Inc. (1976). It has been generalized in the WHAMO model to include the inner equations pertaining to any element which is part of a serial branch.

The method of forward construction is to go from internal node to internal node in a branch, beginning with the second internal node from the upstream end, determining at each node forward construction equations expressing H and Q at that node as a linear function of H and Q at the upstream end of the branch. The form of the forward construction equations is:

$$H(I) = CH1(I) * H(1) + CH2(I) * Q(1) + CH3(I) \tag{6.98}$$

$$Q(I) = CQ1(I) * H(1) + CQ2(I) * Q(1) + CQ3(I) \tag{6.99}$$

where H(I) is the energy head at node i
Q(I) is the discharge at node i
CQ1, CQ2, CQ3, CH1, CH2, and CH3 are computed forward construc -tion coefficients
Node 1 is the upstream node of the branch

Clearly, the forward construction equations for the downstream node of a branch are equivalent to the reduced outer equations for that branch.

The process of determining equations (6.98) and (6.99) is a recursive process in which the forward construction coefficients CQ1, CQ2, ..., CH3 at node i are calculated for those at the preceding node i-1 according to the inner equations written between the two node points. A separate description follows for the different forms of inner equations which apply to the various element types.

6.9.1 Forward Construction for Conduits

The conduit inner equations of Section 6.2 can be rearranged algebraically to the form:

$$H(I) = E(I) * H(I-1) + F(I) * Q(I-1) + G(I) \quad \textbf{(6.100)}$$

$$Q(I) = R(I) * H(I-1) + S(I) * Q(I-1) + T(I) \quad \textbf{(6.101)}$$

where Coefficients E, F, ..., T relate directly to α, β, γ, and δ of equations (6.7) and (6.8) as shown in Appendix A of Resource Analysis (1976).

Using these inner equations and the forward construction equations, the recursion relations for determining the forward construction coefficients at node I from those at node I-1 can be developed as follows:

$$H(I) = E(I) \ x \ H(I-1) + F(I) \ x \ Q(I-1) + G(I) \quad \textbf{(6.102)}$$

$$= E(I) \ x \ [CH1(I-1) \ x \ H(1) + CH2(I-1) \ x \ Q(1) + CH3(I-1)] +$$

$$F(I) \ x \ [CQ1(I-1) \ x \ H(1) + CQ2(I-1) \ x \ Q(1) + CQ3(I-1)] + G(I)$$

$$= [E(I) \ x \ CH1(I-1) + F(I) \ x \ CQ1(I-1)] \ x \ H(1) +$$

$$[E(I) \ x \ CH2(I-1) + F(I) \ x \ CQ2(I-1)] \ x \ Q(1) +$$

$$E(I) \ x \ CH3(I-1) + F(I) \ x \ CQ3(I-1) + G(1)$$

Therefore, from equations (6.98) and (6.102), it can be seen that:

$$CH1(I) = E(I) \ x \ CH1(I-1) + F(I) \ x \ CQ1(I-1) \quad \textbf{(6.103)}$$

$$CH2(I) = E(I) \ x \ CH2(I-1) + F(I) \ x \ CQ2(I-1)$$

$$CH3(I) = E(I) \times CH3(I-1) + F(I) \times CQ3(I-1) + G(I)$$

In a similar manner, the following recursion relations for CQ1, CQ2, and CQ3 and be developed:

$$CQ1(I) = R(I) \times CH1(I-1) + S(I) \times CQ1(I-1) \quad \textbf{(6.104)}$$

$$CQ2(I) = R(I) \times CH2(I-1) + S(I) \times CQ2(I-1)$$

$$CQ3(I) = R(I) \times CH3(I-1) + S(I) \times CQ3(I-1) + T(I)$$

Inspection of the forward construction equations (6.98, 6.99), reveals that in every case, no matter what type of element follows, the coefficients for the first node of a branch are:

$$\begin{aligned} CH1(1) &= 1 \\ CH2(1) &= 0 \\ CH3(1) &= 0 \\ CQ1(1) &= 0 \\ CQ2(1) &= 1 \\ CQ3(1) &= 0 \end{aligned} \quad \textbf{(6.105)}$$

The above is simply a statement of the identity principle.

6.9.2 Forward Construction for Turbo-Machine and Valves

The inner equations which pertain to turbo-machines and valves, including ONEWAY and RELIEF elements, can be written in the general form:

$$H(I) = H(I-1) + SLOPE \times Q(I-1) + CONST \quad \textit{(see Eq 6.17)}$$

$$Q(I) = Q(I-1) \quad \textit{(see Eq 6.18)}$$

From the above equations it is easy to determine the following formulae for the forward construction coefficients at node I on the downstream side of a machine or valve:

$$CH1(I) = CH1(I-1) + SLOPE \; x \; CQ1(I-1) \tag{6.106}$$

$$CH2(I) = CH2(I-1) + SLOPE \; x \; CQ2(I-1)$$

$$CH3(I) = CH3(I-1) + SLOPE \; x \; CQ3(I-1) + CONST$$

$$CQ1(I) = CQ1(I-1)$$

$$CQ2(I) = CQ2(I-1)$$

$$CQ3(I) = CQ3(I-1)$$

The above relations can also be employed for diameter changes and dummy elements by replacing

$$SLOPE = 0$$

$$CONST = -HL$$

where HL = calculated head loss.

6.9.3 Forward Construction Including Boundary Conditions

Serial branches which end or begin with boundary conditions receive special treatment in order to eliminate the known variable from the matrix.

This will be illustrated with the simplest case, a one branch two node system. The equations for this system are:

$$H(1) = HBC$$

$$H(2) = CH1(2) * H(1) + CH2(2) * Q(1) + CH3(2)$$

$$Q(2) = QBC$$

$$Q(2) = CQ1(2) * H(1) + CQ2(2) * Q(1) + CQ3(2)$$

In matrix form:

$$\begin{bmatrix} 1 & 0 & 0 & 0 \\ CH1(2) & CH2(2) & -1 & 0 \\ CQ1(2) & CQ2(2) & 0 & 0 \\ 0 & 0 & 0 & 1 \end{bmatrix} \begin{Bmatrix} H(1) \\ Q(1) \\ H(2) \\ Q(2) \end{Bmatrix} = \begin{Bmatrix} HBC \\ -CH3(2) \\ -CQ3(2) \\ QBC \end{Bmatrix} \qquad (6.107)$$

with a known head at node 1 being HBC and a known Q at node 2 being QBC. The variables H(1) and Q(2) can be eliminated from the above matrix as these are known quantities. The four equation system is thus reduced to the two equation-two unknown system shown below:

$$\begin{bmatrix} CH2(2) & -1 \\ CQ(2) & 0 \end{bmatrix} \begin{Bmatrix} Q(1) \\ H(2) \end{Bmatrix} = \begin{Bmatrix} -CH3(2) - CH1(2)\ x\ HBC \\ -CQ3(2) - CQ1(2)\ x\ HBC + QBC \end{Bmatrix} \qquad (6.108)$$

This elimination is done in the program for all reservoirs and flow boundary conditions.

6.9.4 Forward Construction for Surge Tanks

Forward construction proceeds to the last node at the water surface of a surge tank in the usual manner described above. Generally, the conduit equations apply. For the first and last segments of a differential surge tank, the forward construction integrates the inner equations (6.61, 6.62) and (6.61, 6.64) of Section 6.6.3.

At the last node, IL, of a surge tank, the forward construction equations are initially of the usual form:

$$H(IL) = CH1(IL)\ x\ H(1) + CH2(IL)\ x\ Q(1) + CH3(IL) \qquad \text{(see Eq 6.98)}$$

$$Q(IL) = CQ1(IL)\ x\ H(1) + CQ2(IL)\ x\ Q(1) + CQ3(IL) \qquad \text{(see Eq 6.99)}$$

Added to these are the boundary equation 6.47 at the last node

$$H(IL) = TN - TA \times Q(IL) \qquad \text{(see Eq 6.47)}$$

The two equations (6.99) and (6.47) are combined, and the variable Q(IL) eliminated, resulting in a single equation which expresses H(IL) in terms of H(1) and Q(1).

$$H(IL) = -TA \times Q1(IL) \times H(1) - TA \times CQ2(IL) \times Q(1)$$

$$- TA \times CQ3(IL) + TN \qquad \textbf{(6.109)}$$

Equations (6.98) and (6.109) are adopted as the outer equations for this branch, thus eliminating one equation and one unknown from the system matrix. The variable Q(IL) is evaluated after solution of the matrix using equation (6.47).

6.9.5 Backsubstitution

The branch reduction process eliminates the necessity of solving for the H and Q at every internal node of the system simultaneously. This is a great computational saving. Nevertheless, those H and Q variables which are not explicitly included in the outer equations still must be evaluated at each time step.

The process is simple. After solution of the system matrix the H and Q at the ends of each branch are known. The values at the intermediate nodes are then calculated from the values at the upstream end using the forward construction equations (6.98, 6.99) and the coefficients which were determined and saved for each node during forward construction.

As a check for computational accuracy, the H and Q at the last node of a branch are calculated using backsubstitution and compared to the values determined from solution of the system matrix. If these do not agree, a warning message is printed.

6.10 Steady State Generation

The strategy of the initial steady state generation is outlined in Section 2.7. In general, the governing equations for the various elements are the same as those presented previously for the transient simulation. The exceptions are conduit and surge tank elements.

For conduits, the steady state governing equations are these:

Momentum: $$H_j - H_{j+1} = SLOPE \ x \ Q_j + CONST \qquad (6.17)$$

Continuity: $$Q_{j+1} = Q_j \qquad (6.18)$$

Compressible and unsteady effects need not be considered in steady state calculations. Only friction losses are included in the momentum equation. Losses are calculated the same as for the transient case.

$$HL = \left(\frac{f\Delta x}{D} + C_{add} \right) \frac{Q|Q|}{2gA^2}$$

This term is then linearized in the manner described for values in Section 6.3.3 using a preliminary estimate of Q generated by the program or an estimate of Q equal to the value calculated at the last iteration. Forward construction proceeds as before using the altered inner equations.

Under steady state conditions there will be no flow in a surge tank and the head at each of the internal nodes contained therein will be equal. This is accomplished by equating the forward construction coefficients at each of the internal nodes of a surge tank.

$$\begin{aligned} CH1(I) &= CH1(I-1) \\ &\cdot \\ &\cdot \\ &\cdot \\ CQ3(I) &= CQ3(I-1) \end{aligned} \qquad (6.111)$$

Forward construction, system solution, and backsubstitution calculation proceeds exactly as for the transient simulation, excepting only the alterations enumerated above for conduits and surge tanks.

6.11 Interpolation

In general, all interpolations performed in WHAMO use the method of continuous parabolic interpolation. A thorough delineation of this technique is contained in Appendix G of Perkins (1964) part II or in the original source, Snyder (1961). The same interpolating polynomial is also used in the linearization of machine characteristics outlined in Section 6.4.

This technique is in general quite satisfactory. An advantage over other techniques is that it provides a continuous first derivative at all points of the interpolating curve. Under certain circumstances, however, particularly where there is an abrupt change of slope implied in the data, the technique has been found to give unreasonable results. A typical example is illustrated in Figure 42. To guard against this situation, a check has been built into the interpolation routine. If four neighboring data points are continuously monotonic, and if the interpolating curve between the two central points of the four is not continuously monotonic, then the continuous parabolic interpolation is replaced by a simple straight line interpolation between those two points.

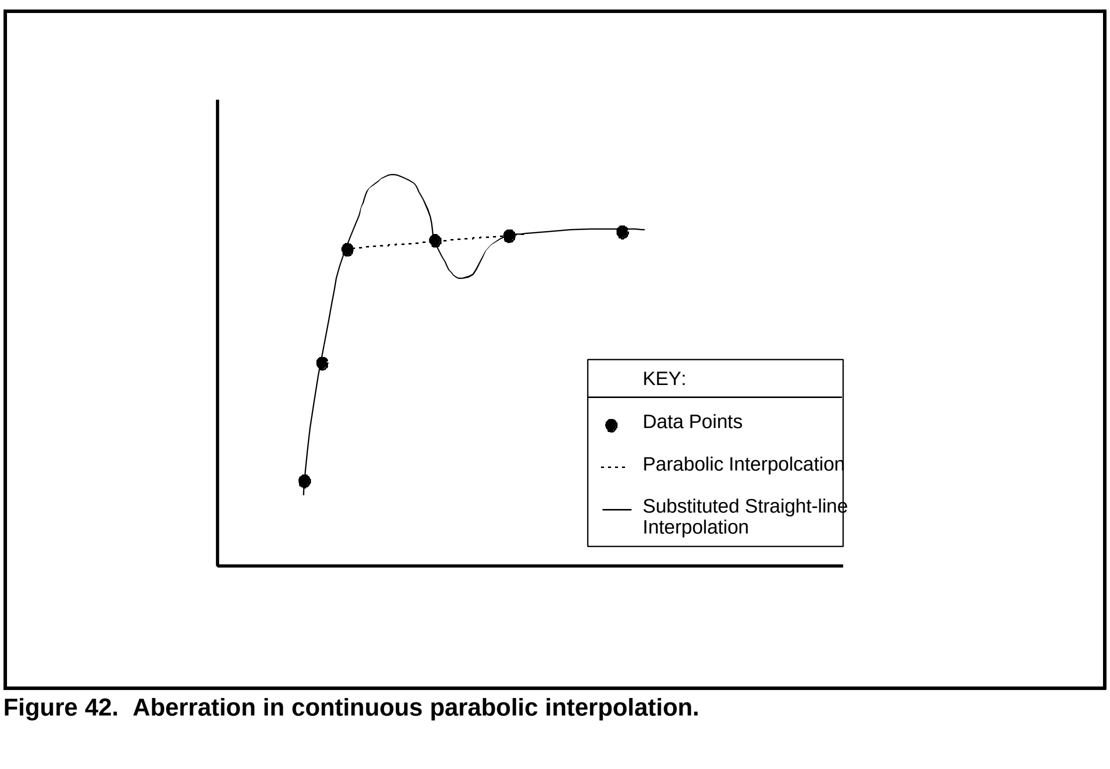


Figure 42. Aberration in continuous parabolic interpolation.

7 Conclusion

WHAMO 2.0 for DOS has been upgraded to WHAMO 3.0 for Windows. WHAMO 3.0 is a valuable design and analysis tool for hydropower plants and fluid distribution systems that are subject to water hammer. WHAMO performs dynamic simulation of networks comprised of pipes, valves, pumps, tanks, and turbomachines and calculates time-varying flows, pressures, heads, and other parameters throughout the network. WHAMO 3.0 utilizes the same time-tested algorithms that have been used in previous versions of WHAMO.

The new WHAMO Network Builder has greatly simplified the data input process. In the past, users had to use a text editor along with a series of complex commands to create the input file. This method of data input still works, but the new Network Builder is easier and more efficient for most users. The module is graphically-based and allows users to “draw” the network on the computer screen and input the relevant data via a series of easy-to-understand dialog boxes.

The new WHAMGR program for display of WHAMO simulation results is also Windows-based. In the past, users were restricted to a few printers and monitors to display output, and output requests had to be entered via complex commands. The new WHAMGR allows users to quickly and easily select, display, and print output on any printer or monitor that is supported by Windows.

References

Carnahan, B., H.A. Luther, and J.O. Wilkes, *Applied Numerical Methods*, New York: John Wiley & Sons, 1969

Crane Co., *Flow of Fluids through Valves, Fittings, and Pipes,* Technical Paper No. 410, New York, New York, 1979

Daily, J.W., "Hydraulic Machinery," in *Engineering Hydraulics,* ed. H. Rouse, New York: John Wiley & Sons, 1976

Gardel, A., Chambre D'Equilibre, F. Rouge, and Cie, Lausanne, 1965

Graze, H.R., "A Rational Thermodynamic Equation for Air Chamber Design," Third Australasian Conference on Hydraulics and Fluid Mechanics, Sydney, 1968

Graze, H.R., J. Schubert, and J.A. Forrest, "Analysis of Field Measurements of Air Chamber Installations," *Proceedings, Second International Conference on Pressure Surges*, BHRA, London, 1976

Hagihara, S., H. Yokota, K. Goda, and K. Isobe, "Stability of a Hydraulic Turbine Generating Unit Controlled by P.I.D. Governor," IEEE Power Engineering Society Winter Meeting, New York, 1979

International Business Machines Corporation, *System/360 Scientific Subroutine Package,* Version III, Programmer's Manual, White Plains, New York, 1970

Knapp, R.T., "Complete Characteristics of Centrifugal Pumps and Their Use in the Prediction of Transient Behavior," *Trans. ASME, Vol. 59*, 1937

Marchal, M., G. Flesch, and P. Suter, "The Calculation of Waterhammer Problems by Means of the Digital Computer," *Proceedings of the International Symposium on Waterhammer in Pumped Storage Projects,* ASME, Chicago, November 1965

Parmakian, J., Waterhammer Analysis, New York: Dover Publications, Inc., 1963

Perkins, F.E., A.C. Tedrow, and P.S. Eagleson, Discussion of Streeter, V.L., and C. Lai, "Water-Hammer Analysis including Fluid Friction," *Journal of the Hydraulics Division*, ASCE, Vol. 89, No. HY1, January 1963

Perkins, F.E., A.C. Tedrow, P.S. Eagleson, and A.T. Ippen, "Hydropower Plant Transients, Part II, Response to Load Rejection," *Hydrodynamics Laboratory Report No. 71*, Massachusetts Institute of Technology, Department of Civil Engineering, September 1964

Perkins, F.E., A.C. Tedrow, P.S. Eagleson, and A.T. Ippen, "Hydropower Plant Transients, Part III, Response to Variable Load," *Hydrodynamics Laboratory Report No. 79*, Massachusetts Institute of Technology, Department of Civil Engineering, September 1965

Resource Analysis, Incorporated, "Model for the Investigation of Hydraulic Transients in Pipe Networks Due to Unsteady Pump/Valve Operation," report to the Omaha District, U.S. Army Corps of Engineers, August 1976

Resource Analysis, Incorporated, "Results of Mathematical Simulations of the Oahe Power and Pumping Plant Hydraulic Transients," report to the Omaha District, U.S. Army Corps of Engineers, February 1978

Snyder, W.M., "Continuous Parabolic Interpolation," *Proc. ASCE, Vol. 87, No. HY4*, July 1961

Waterways Experiment Station, "Graphics Compatibility System (GCS) Programmers Reference Manual," Computer Program No. 803-F3-R0200, U.S. Army Corps of Engineers, Vicksburg, Mississippi, 1979

Waterways Experiment Station, "Hydraulic Design Criteria," U.S. Army Corps of Engineers, Vicksburg, Mississippi, January 1977

Wylie, E.B., and V.L. Streeter, Fluid Transients, New York: McGraw-Hill Inc., 1978

Appendix: Index of Primary Commands in WHAMO 3.0