SOLUTION TO FLOW PROBLEM
IN SYSTEMS OF PIPES

Johann H. Lang

A DISSERTATION

IN THE

FACULTY OF ENGINEERING

Presented in partial fulfilment of the requirements for the

Degree of MASTER OF ENGINEERING

at

Sir George Williams University

Montreal, Canada

Date: March 1972

ABSTRACT

A method for solving pressure and flow problems of systems of pipe lines is developed.

Based on this analysis a computer program is developed, which permits the solution to be obtained with ease.

The program utilizes the latest developments to determine loss coefficients for use on the analysis.

ACKNOWLEDGMENT

The author wishes to thank Sir George Williams University for conceiving the academic program which makes it possible for practicing engineers to renew their knowledge and broaden their horizons.

In particular the author wishes to thank Professor M. P. du Plessis, Ph.D., Chairman, Department of Mechanical Engineering for his guidance in this project.

Above all I wish to express my gratitude and thanks to my family, especially my wife Wilma Jean and my children Steven and Susanne who for the last years have accepted a part time husband and father.

INDEX

I	Summary	1
II	Introduction	3
III	General Problem	
III.- 1	Basic Equations for Fluid Flow	6
III.- 2	Terminology	14
III.- 3	Mathematical Formulation	19
III.- 4	Outline of Solution	28
IV.	Calculation of Loss-coefficients	
IV.- 1	General	30
IV.- 2	Conduits	33
IV.- 3	Nodes	44

INDEX (contd.)

V.	Computer Program	
	General	62
V.- 1	Data Input	63
V.- 2	Checking of data	64
V.- 3	Printing of Input data	65
V.- 4	Initialization	66
V.- 5	Iterative Solution	67
VI	Test Examples and Discussion	68
References		71

List of Figures

Figure	1	Typical part of System
Figure	2	Equation (3.7)
Figure	3	System and System Matrix
Figure	4	Determination of Q"
Figure	5	Definitions for Pipe connections
Figure	6	Definitions for Y-Branches
Figure	7	MACRO-Flow Chart
Figure	8	Schematic of Examples
Figure	9	Results of computations

List of Tables

Table 4.1 Equivalent Sand-roughness

Table 4.2 Valve Performances

Table 4.3 Turbine performance

I. SUMMARY

As part of a study of unsteady flow calculations, a method for the solution of steady state flow was developed and is presented here. While no startling new discoveries can be claimed, it is believed that this work constitutes a departure from the solutions thus far proposed by including the so called minor losses into the solution.
Furthermore the work constitutes an attempt to compile systematically existing current knowledge on steady state flow losses through conduits and elements of conduit networks.
A Computer program based on this method and the data given, was developed. Emphasis is placed on simplifying the use of the program so that it can be used used without special training by engineers confronted with a particular problem.
While the program in the form developed sofar favors hydropower systems it is by no means confined to that specific field. Any special configuration can be programmed as a subroutine and made part of the program. However, the analysis and the programs are at the moment restricted to closed conduits (channel flow is excluded).
Loss coefficients for the following pipe system elements are included in the current computer program.

Conduits	Round pipes	constant area
	Non-round conduits,	constant area
	Round pipes	changing area

	Pipe bends	constant area
Nodes	Reservoir	
	Pipe intake	
	Pipe connection	
	Pipe elbow	
	Y-branches, Tees	
	Valves	
	Turbo-machinery	

The coefficients presented are based on data available currently in literature.

An overview only of the computer program mentioned earlier is given.

A small sample problem run with the program is presented, details of input and the printed results are shown in Appendix A.

There is no Nomenclature given in this dissertation. Symbols used are explained wherever they first appear.

II. INTRODUCTION

During the implementation of the unsteady flow solution proposed and outlined by Streeter [16], the need for a method determining initial flow conditions with reasonable accuracy became quite apparent.

> The method of characteristics used to solve the unsteady, one-dimensional equations of conduit flow requires the initial flow as the starting point for subsequent calculations.
> These initial conditions must conform to the mathematical modelling of losses used, otherwise an artificial change in pressure is calculated for each of the non-conforming conditions, which propogates through the system falsifying the computed results.

The problem of determining steady state flow within a system of conduits is quite old. It assumed particular importance with industrialization and urban planning. The earliest modern work in this field is generally conceded to be due to Cross [14].

> Cross assumes that the friction losses in pipes are dominant and thus justifies the deletion of losses in junction etc. Inclusion of any junction losses in the calculation must be in the form of an equivalent pipeloss. The losses in the pipes are assumed to follow the well known

Hazen-Williams equation. The method is limited due to these restrictions.

Cross published his analysis in 1934, it was not until the early 1960's that further investigation into the problem were conducted, with publications as recent as February 1972 (Ref.[17],[18]).

All of the methods published during this period are using large high speed computers and most are written, utilizing system methods such as graph-theory in the automatic determination of the relevant coefficients for the equations.

In all of these recent publications emphasis is placed on the methods for the mechanical determination of the coefficients for the equations as well as on the methods used to solve the resulting system of non-linear equations.

However, in most of the recent work, the concepts of the actual flow in the pipes developed by Cross are accepted. No attempt is made to incorporate some of the more recent works on resistance factors of conduit components.

The newer methods then, suffer from the same restrictions as the original Cross-method, restrictions which may not be too severe when faced with pure water distribution net works, but which in other cases do not permit using the methods.

wider application of the methods requires improvements in the following areas :

a) Incorporate recent developments of turbulent pipe flow work into the analysis

b) Include losses other than pure pipe friction losses into the analysis.

The first requirement can easily be fullfilled within the framework of any of the more recently developed methods. However, the second reqqirement necessitates considerable modification in the definitions of a "circuit" used to establish the relevant equations.

For this reason the work described in the following sections does not utilize directly the formal approach of the graph theory in formulating the relevant equations.

Instead each conduit and each node are separately described with appropriate equations, the equations are assembled into a system matrix with non-linear terms and this is solved utilizing a linearization process for each iteration.

In order to redefine the appropriate relations as well as to help in clarifying the proper use of loss coefficients, the relevant equations are traced from the general equation of motion in the following section.

III. GENERAL PROBLEM:

III.- 1) Basic equations for fluid flow.

The equation of motion (momentum equation) in Cartesian tensor notation (*) for incompressible fluid flow is :

$$(3.1) \qquad \underbrace{\frac{\partial \overline{v_i}}{\partial t} + \overline{v_j}\,\overline{v_{i,j}} = f_i - \frac{1}{\rho}\overline{p}_{,i} + \nu\,\overline{v_{i,jj}}}_{\text{MEAN FLOW EQUATION (Navier - Stokes)}} - \underbrace{(\overline{v_i' v_j'})_{,j}}_{\text{REYNOLDS STRESS}}$$

The continuity equation is:

$$(3.2) \qquad \overline{v_{i,i}} = v'_{i,i} = 0$$

In both equations the bar ($\overline{v}$) denotes time averaged values and the dash (v') represents the time varying (turbulent) component. In addition the following definitions hold :

- v fluid velocities
- f body forces

(*) Familiarity with the notations of tensor algebra is assumed. (see Reference [15]

p	fluid pressures
ρ	density of fluid
ν	kinematic viscosity

Equation 3.1 can be re-written as:

$$\frac{\partial \bar{v}_i}{\partial t} + \bar{v}_j \bar{v}_{i,j} - f_i + \frac{1}{\rho}\bar{p}_{,i} = \nu \overline{v_{i,jj}} - (\overline{v_i' v_j'})_{,j} \qquad (3.3)$$

The following assumptions and definitions are now made:

Steady (time invariant) flow $\frac{\partial \bar{v}_i}{\partial t} = 0$

Vorticity Vector $\xi_i = \epsilon_{ijk} v_{k,j} = (v_{k,j} - v_{j,k})$

Conservative body forces $f_i = -\phi_{,i}$

$$p_i = \frac{d}{dx_i}\int \frac{dp}{\rho} = P_{,i}$$

After some manipulations, the left hand side of equation 3.3 can be rewritten as:

a) $$\epsilon_{ijk} v_j \xi_k + \left(\frac{1}{2} \bar{v}_j \bar{v}_j + \phi + P \right)_{,i}$$

Making the additional, assumption that the mean flow is irrotational it can then be seen that the left hand side of equation (3.3) can be reduced to the well known Bernoulli function along a stream tube for this specific case.

While there has been and still is a great deal of effort exerted to obtain usable expressions for the Reynolds stresses, one of the more useful concepts is due to Prandtl who expanded the ideas advanced by Boussinesque (from Schlichting [1]).

Prandtl's mixing length theory essentially replaces the expression for the Reynolds stress as follows:

$$-\overline{v_i' v_j'} \simeq \frac{1}{\rho}\overline{v_{i,j}}\,\varepsilon$$

with $\varepsilon = l^2\left|\frac{du}{dy}\right|$

where l is a characteristic length

This approximation has proven to be extremely valuable and has been applied to or forms the basis for most of the detailed solutions for Boundary Layer Flow. By its very concept it is a function intimately related to the flow field rather than being a property of the fluid. This must be kept in mind when applying experimental turbulent flow data from one configuration to another.Also the apparent viscosity factor is zero whenever the gradient is zero (maximum and minimum of velocity). This defect is overcome by Van Karman's similarity hypothesis, which relates the Reynolds-stress to a combination of first and second velocity derivatives. A number of other more recent developments are also being used for detailed flow analysis.

However, for the purpose of this derivation only the concept that the Reynolds-stress can be related to the local velocity gradient is of importance. This will be clear in the further development.

With this concept the right hand side of equation (3.3) can be expressed as follows:

b) $$\rho\nu\,\overline{v_{i,jj}} - \rho(v_i'v_j')_{,j} = (\mu+\varepsilon)\,\overline{v_{i,jj}}$$

Using equations (a) and (b), equation (3.3) is rewritten as follows:

(3.4) $$\rho\left(\tfrac{1}{2}\overline{U}^2 + \phi + P\right)_{,i} = (\mu+\varepsilon)\,\overline{v_{i,jj}}$$

where U^2, the velocity vector replaces the term $\overline{v_j}\,\overline{v_j}$

In the case of inviscid fluid flow the R.H. side of the above equation is zero and integration gives the well-known Bernoulli equation.

It should be noted that in equation (3.4) the apparent stress coefficient is assumed to be a constant , an obvious contradiction of its original derivation. It is also understood that in the presence of turbulent flow a number of the assumptions leading to this result are tenuous to say the least.(steady flow, absence of rotation,etc.)

Relatively simple integration of equation (3.4) is only possible in the case of laminar pipe flow, which immediately leads to the parabolic velocity distribution of the Hagen-Poiseuille equation.

For more complicated configurations as well as for turbulent pipe flow a modification of equation (3.4) is used.

Since in most cases of conduit flow the local velocity distribution is of no great concern, the pipe flow is envisaged to be uniform across the section with the shear force acting as surface or traction force at the wall of the conduit (plug flow). Since this now assumes an infinite gradient at the wall, the complete right hand side of equation (3.4) is replaced by:

$$(\mu + \varepsilon)\overline{v_{i,jj}} \simeq -fU^2$$

The factor f thus defined, is a function of the velocity, cross section, typical dimensions of the conduit and especially the wall conditions as well as the fluid viscosity.

Using the above expression in equation (3.4) and since these assumptions imply one dimensional flow, equation (3.4) can then be integrated to give:

$$(3.5) \qquad \rho\left(\tfrac{1}{2}U^2+\phi+P\right)_{x=L}-\rho\left(\tfrac{1}{2}U^2+()+P\right)_{x=0}=-fLU^2$$

This basic equation, which is essentially a modified form of the inviscid Bernoulli equation has been applied to many engineering problems. It will obviously result in reasonable approximations if the loss factor f on the right hand side is known or can be inferred from previous data.

The utility of this solution, developed to cover flow in conduits only, extends to flows across valves, orifices and other similar types.

The factor f is determined on an experimental basis with the experimental data presented in terms of U^2, with U being a typical velocity in the system.

Equation (3.5) with some modifications forms the basis for this analysis. In section IV, loss coefficients for various conduits and connections are presented.

For all of these coefficients, particularly wherever sufficient data is available, the dependence of f on factors such as Reynolds-Number, can very clearly be seen, and with the method outlined further on in this chapter, this can be considered in the analysis.

In order to complete the derivation of the equation used, equation (3.5) is rewritten.

Subscript 1 refers to location x = 0

Subscript 2 refers to location x = L.

$$(P_1 - P_2) + \rho(\phi_1 - \phi_2) + \frac{\rho}{2}(U_1^2 - U_2^2 + \frac{2}{\rho} f L U_r^2) = 0$$

with the assumptions leading to equation (3.5), U_1, U_2 and U_r, can be replaced by Q/A_1, Q/A_2 and Q/A_r. (A_r is in this case the reference for which f is valid).

By definition the conservative body force potential is a function of the location only. In the case of gravity force it is simply the difference in elevation between end 1 and end 2 of the conduit multiplied by the value for g. With these simplifications the following is obtained:

$$(P_1 - P_2) + \rho g(El_1 - El_2) - \frac{\rho}{2} Q^2 (\frac{1}{A_2^2} - \frac{1}{A_1^2} + \frac{2}{\rho A_r^2} f L) = 0$$

The last term is combined into the loss-cefficient C and the above equation becomes:

(3.6) $$P_1 - P_2 - C\,Q^2 = \rho g(El_2 - El_1)$$

The term "loss coefficient" used above is actually a misnomer and may be misleading. A better term would be "velocity effect coefficient". However, since the quantity as defined above is encountered in literature very often and defined as loss coefficient, it will continue to be called this, with the understanding that it includes both, true loss and the effect of the velocity on pressure.

III.- 2) Terminology:

In order to permit definition of the problem in an orderly fashion, the terminology described below will be followed: (see Figure I)

CONDUIT This is a part of the system that throughout its length has uniform properties inasmuch as they affect the flow.

Pipe of uniform cross section

Conduit of any other shape

The positive direction of flow is determined by defining the two ends of the conduit as end 1 and end 2.

Flow is positive when flowing from end 1 to end 2.

Each conduit is described with one equation (based on equation (3.5)).

$$P_{i,1} - P_{i,2} - C_{i,Q}\, Q_i|Q_i| = \gamma(El_{i,2} - El_{i,1}) \qquad (3.7)$$

$P_{i,1}$ pressure in conduit i at end 1.

$$\left[\frac{\text{unit of force}}{\text{unit of area}}\right] \quad (*)$$

$P_{i,2}$ pressure in conduit i at end 2

Q_i flow through conduit i

$$\left[\frac{\text{unit of volume}}{\text{sec}}\right]$$

$C_{i,Q}$ loss - coefficient for conduit i for the given flow Q, including the terms for the kinetic energy.

$$\left[\frac{\text{unit of force x sec}^2}{\text{unit of area x (unit of volume)}^2}\right]$$

$El_{i,\ldots}$ Elevation of ends

$$\left[\text{unit of length}\right]$$

As can be seen there are three unknowns for each conduit. A given system will thus result in:

3 x N unknowns

N...number of conduits.

(*) In defining the various quantities used, rather than be confined to a given system of units, definitions of dimensions will be used throughout as given above.

NODES These are the parts of the system connecting conduits with each other or with the outside. These nodes represent physically anything from a single pipe connection to a pump or turbine.Their main characteristic is the fact that the pressure change for a given flow can be described with a simple lumped parameter.Each node may be described in general terms as to its type as shown below, but because each node connects a unique set of conduits, each node is a unique entit .

The nodes may in general be divided into three groups:

End Node

Through Node

Branch Node

Nodes are described,depending on type,with an energy equation similiar to (3.6) and a continuity equation. Equation (3.6) is further simplified since the difference in elevation is zero.

Depending on the general type of node the following equations are obtained.

(3.8) $$P_{i,1 or 2} + C_{n,Q} Q_i |Q_i| = \rho g S$$ End Node

(3.9) $P_{i,2} - P_{j,1} - C_{n,i} Q_i|Q_i| = 0$ Through Node

(3.10) $Q_i - Q_j = 0$

(3.11) $P_{i,2} - P_{j,1} - C_{n,Q} Q_i|Q_i| = 0$ Branch Node

(3.12) $P_{i,2} - P_{j,1} - C_{n,Q} Q_k|Q_k| = 0$

(3.13) $Q_i - Q_j - Q_k = 0$

Subscripts : i, j, k, refer to the conduits connected by node n.

n, refers to the node n

C loss coefficient for node n and for the flow identified with it. (dimensions as for conduit).

S (in equation (3.8).
This is the submergence of the conduit centerline in feet.
In the case of a pressure chamber this includes any gas pressure on the free surface

Since the nodes are not adding any unknowns to the system of equations, it can readily be shown that the equations for the nodes together with the ones for the conduits, completely describe the system and thus must result in a unique solution.

III.- 3) Mathematical Formulation:

In the preceding paragraph it was shown that flow through a system of conduits and nodes can be described by a set of equations:

Conduit	i	$P_{i,1} - P_{i,2} - C_{i,Q}\,Q_i\lvert Q_i\rvert = \rho g(El_{i,2} - El_{i,1})$	(3.7)
End Node	j	$P_{e,1} + C_{j,Q}\,Q_e\lvert Q_e\rvert = P_{UP}$	(3.8a)
		OR	
		$P_{e,2} + C_{j,Q}\,Q_e\lvert Q_e\rvert = P_{DOWN}$	(3.8b)
Through node	k	$P_{e,2} - P_{f,1} \quad C_{k,Q}\,Q_f\lvert Q_f\rvert = 0$	(3.9)
		$Q_e - Q_f = 0$	(3.10)
Branch node	m	$P_{e,2} - P_{f,1} - C_{m,Q}Q_f\lvert Q_f\rvert = 0$	(3.11)
		$P_{e,2} - P_{g,1} - C_{m,Q}Q_g\lvert Q_g\rvert = 0$	(3.12)
		$Q_e - Q_f - Q_g = 0$	(3.13)

Subscripts e, f, g, refer to the conduits connected to the respective nodes.

It was noted earlier on that the above equations are sufficient to determine the number of unknowns generated in describing the system. However one is faced with the fact that the flows Q appear as a linear term as well as a quadratic term, thus rendering the set of equations describing the system non-linear.

> The theory of non linear systems of equations is at this point of time not as a advanced as the theories for solving linear equations.
>
> In any case, a non-linear system of equations can today only be solved by iteration methods. However, generalized methods applicable to any type of non-linear systems do not necessarily converge to a solution, their convergence is usually dependent upon the initial starting point for the iteration [2].

The method proposed here for the solution of this particular type of system is a combination of the method of false position and the derivative method (Jacobian).

Figure 2 shows a typical equation (3.7). It should be noted here, that the use of $Q|Q|$ in equation (3.7) results in the physically correct picture shown in Figure 2, where the left branch of the parabola is inverted.

It is obvious that in the neighbourhood of Q the quadratic equation (3.7) can be replaced by a linear equation of the form:

(3.14) $$P_{i,1} - P^{*}_{i,2} + m_i Q_i = \rho g (H_{i,2} - H_{i,1}) - b_i$$

* asterisk on P means approximated by linear equation.

m_i slope of line A as shown in Figure 2

b_i a constant due to approximation.

The values of m and b are calculated for the current value of Q . It can easily be shown that the following is valid.

(3.15) $$m_i = \frac{C_i}{2\Delta Q}\left[(Q_i - \Delta Q)\,|Q_i - \Delta Q| - (Q_i + \Delta Q)\,|Q + \Delta Q|\right]$$

(3.16) $$b_i' = -Q_i\,(m_i + C_i\,|Q_i|)$$

$$b'' = -(Q_i + \Delta Q)(m_i + C_i|Q_i + \Delta Q|)$$

$$b_i = \tfrac{1}{2}(b' + b'')$$

ΔQ flow difference, arbitrarily selected

Equation (3.15) is obtained by solving equation (3.7) for $P_{i,2}$ using $(Q + \Delta Q)$ and $(Q - \Delta Q)$, equation (3.16) is obtained by equating equations (3.7) and (3.14)

At this point of the development some comments are required as to why the local derivative is not used to linearize equation (3.7)

Since one must expect that during

> iteration some of the quantities will be zero, -as a matter of fact this is the point of starting all iterations, - and since the derivative at zero flow is also zero, the system of equations would become singular and thus no solution would be possible.

Returning to the beginning of this section where the equations for the system were described, the equations containing the quadratic term are replaced by linearized equations as shown earlier.

(3.17) $P_{i,1} - P_{i,2} + m_i Q_i = \rho g(El_2 - El_1) - b_i = B_i$ Conduit

(3.18a) $P_{e,1} + m_j Q_e = P_{UP} - b_j = B_j$ End Node

OR

(3.18b) $P_{e,2} + m_j Q_e = P_{DO} - b_j = B_j$

(3.19) $P_{e,2} - P_{f,1} + m_k Q_f = -b_k = B_k$ Through Node

(3.20) $Q_e - Q_f = 0$

(3.21) $P_{e,2} - P_{f,1} + m_m Q_f = -b_m = B_m$ Branch Node

(3.22) $P_{e,2} - P_{g,1} + m_n Q_f = -b_m = B_m$

(3.23) $Q_e - Q_f - Q_g = 0$

Subscripts are as explained earlier.

The above equations are now assembled into a system of linear equations. The method used is simply:

1) Assemble all conduits in sequence.

2) Assemble all nodes in sequence.

This assembly results in a matrix 3N x 3N, where N is the number of conduits. In matrix notation this is written as:

$$[\ A\ ]\ [X] = [B] \qquad (3.24)$$

This is solved in the usual manner (see Ref.[2]) for each successive iteration.

The make-up of the matrix A is demonstrated in Figure 3 for the system shown. It is obvious from this example that the system matrix will always be a scarcely populated matrix. Furthermore only the values of m and B will change in successive iterations. The temptation is therefore very strong to solve this system by some of the iteration methods mentioned in the Ref.[2]. However, as mentioned earlier the system may become nonconvergent if the wrong initial vector is used.

Before finalizing the procedure to be used three areas remain to be discussed,

one is the choice of ΔQ

the other is the choice of Q" in subsequent iterations
and the last is the termination of the iterations.

The choice of ΔQ affects the accuracy of any solution of the linearized system. If too big the system is insensitive to the local changes, if too small the system may become unstable in successive iterations. In general the initial value of ΔQ will depend on the particular problem. It must be supplied as initial data for the program.

A reasonable choice has proven to be approximately 2% to 5% of the largest individual flow expected in any particular conduit.

Once the problem is solved for the given Q, subsequent iterations are performed at smaller values as outlined later.

The choice of Q" to be used in the successive linearization procedure can be visualized from Fig. 4:

The physical meaning of the solution to any one of the equations (3.7) to (3.13), as well as (3.17) to (3.23) is,that the pressures are to be balanced.

Having replaced the original

quadratic equation by a linear relation does not alter this basic requirement.

With this in mind a better approximation for Q is to be obtained.

The linear and the quadratic expression for a conduit are:

linear $$P_{i,1} - P_{i,2} + m_i Q_i' = \rho g(El_2 - El_1)_i - b_i$$

quadratic $$P_{i,1} - P_{i,2} - C_i Q_i'' |Q_i''| = \rho g(El_2 - El_1)_i$$

Implicit in the above is the assumption that the correct solution is represented by that flow,which produces the required pressure difference $(P_1 - P_2)$ in the quadratic relation.

Solving for Q" gives:

(3.25) $$H = -\frac{1}{C_i}(m_i Q_i' + b_i)$$

(3.26) $$(\text{sign } H) = \frac{|H|}{H}$$

(3.27) $$Q'' = (\text{sign } H)\sqrt{|H|}$$

The remaining question as to when to stop searching for closer approximation to the true solutions has two answers:

answers:

1) For a given selected ΔQ

A solution for this particular ΔQ has been obtained whenever the differences between all Q(i) and Q(i+1) (current solution and previous estimate) are smaller than the given ΔQ.

As will be seen later, this criterium is also used when updating the linearized coefficients B and m.

2) Determine need for new ΔQ

The criteria used here is the following:

An error F is determined using the current values of the unknowns and solving each quadratic conduit equation. If in any one of the equations the absolute error is greater than .1% of the smallest absolute value of one of the two pressures occurring in the equation, a new ΔQ is selected and this new ΔQ is half the previous value. Whenever all solutions are within the indicated error the problem is solved.

III.- 4) Outline of Solution:

The problem solving part of the computer program outlined in Section V follows the procedure outlined below:

The assumption here is that the values for the loss coefficients are either known or can be determined by the methods outlined in Section IV. Also known are the ways each node connects the conduits.

1) Set up system matrix A.

a) All conduit equations

Designate the unknown values as:

P(i,1)	X(3i-2)
P(i,2)	X(3i-1)
Q(i)	X(3i)

where "i" is the conduit number .

Linearize the equations as outlined earlier.

The matrix coefficients thus determined are:

A(i,3i-2) A(i,3i-1) A(i,3i)

b) All node equations.

Keep a continuous count of the equations.

Linearize as outlined. (Determine the column for the coefficients from the identification numbers of the connecting conduits.)

2) Solve the system of linear equation.

3) Check if solutions of $Q_i = X(3i)$ fall within $\pm \Delta Q$ of the previously assumed solution. Replace value of Q_i with new estimate when required.
If all solutions are within the range indicated continue at at step 4, if not, start with Step 1 again.

4) Solve individual quadratic conduit equations with the last linear solution.Check if error is within prescribed limit for each equation. If not, change ΔQ and start with Step 1 again, if correct continue at Step 5.

5) Solution obtained.

IV.. CALCULATION OF LOSS COEFFICIENTS:

IV.- 1. General

It will obviously not be feasible to describe in a general way all possible configurations expected in a system of pipes. This analysis will show a number of most commonly used configurations. These configurations will be part of the standard computer program described later on.
In addition the program permits future expansion into other configurations, a feature which is described later. Such an expansion can either be in the form of mathematical relations or in the form of tables (empirical data), in any case these are handled as subroutines by the program.

In general the loss coefficients thus calculated must be suitable for use in an equation of the form of equations (3.8), (3.9), (3.11) or (3.12). The numerical value of the loss coefficient returned from such subroutines must have the dimensions:

$$\left[\frac{(\text{unit of force})\ (\text{unit of time})^2}{(\text{unit of area})\ (\text{unit of flow})^2}\right] \tag{4.1}$$

The loss coefficients included in the present analysis are:

Conduits	uniform cross section	round pipes
	arbitrary cross section	
	non uniform cross section diffusors,etc.	

pipe bends

Nodes Entrance and exit from reservoirs

Pipe connections (sudden change in area)

Pipe elbows

Y - Branches (Tees)

Valves (Butterfly, spherical valve, etc.)

Hydraulic turbo-machinery

(turbine & pumps)

What has been done in the above classification is to rank commercial pipe fittings with nodes, whereas pipe configurations (such as bends) encountered in large hydro installations will be treated as conduits.

The designation of Hydraulic Machinery as nodes requires some explanation.

> While such machines are made up of conduits of sometimes considerable length, machine performance is normally known and given as a relation between pressure at two points and the flow. We thus can treat the given machine as a pressure drop between the two points defined, which is nothing else but the definition of a node.

If the analysis presented in the first part of this project is used by itself the notion of conduits and nodes, except for branches, appears to be superfluous. However, since the analysis will later be used to determine transients in pipe lines, this distinction is necessary.

IV.- 2. CONDUITS:

IV.- 2.1 Round Pipes:

This is probably the one area in this field most thoroughly documented. Both theoretical derivations of the wall shear stresses and losses, as well as numerous publications of empirical data are available.

The laminar flow through a pipe was solved very early and the results are the famous Hagen-Poisseuille equation derived independently by these two workers in 1838 and 1846 respectively. The relations developed represent one of the few cases of an exact solution of the Navier - Stokes equations.

A summary of the history of the more important and lasting investigations into turbulent pipe flow is given by Schlichting [Ref.1,p.594].

Prandtl [Ref.3] extended the work of Blasius [Ref.4] and arrived at a universal friction law for smooth pipes (fully developed flow).

(4.2) $$\frac{1}{\sqrt{\lambda}} = 2.\ \log(Re\sqrt{\lambda}) - 0.8$$

Th.van Karman [Ref.5] derived a relation for completely rough pipes, which was modified to fit Nikuradse's [Ref.6] experimental results.

(4.3) $$\frac{1}{\sqrt{\lambda}} = 2.\ \log(\frac{r}{k_s}) + 1.74$$

Equation (4.2) shows obviously no effect of wall roughness, whereas in equation (4.3) no effect can be seen due to Reynolds number.

> There are two regimes and it can be shown that the transition from hydraulically smooth to fully rough is a function of the friction velocity or the wall shear stress itself. (This is caused by the Reynolds number or velocity dependency of the laminar sublayer thickness within the pipe).

The existence of this transition can be shown analytically and it was experimentally confirmed by Nikuradse [Ref.6] as well as in the data for commercial pipes by Moody [Ref.7]. Colebrook and White [Ref.8] established a mathematical correlation for the resistance factor in this zone.

(4.4) $$\frac{1}{\sqrt{\lambda}} = 1.74 - 2.\ \log(\frac{k_s}{r} + \frac{18.7}{Re\sqrt{\lambda}})$$

> It can be seen that for $k_s = 0$ equation (4.4) becomes equation (4.2) and as the Reynolds-number grows large, it approaches equation (4.3).

The Colebrooke function given above fits the data on commercial pipes published by Moody [Ref. 7] . The difference between the values calculated with equation (4.3) and the published data in the fully rough region are minute. For this reason equation (4.4) will be used for the complete range of turbulent pipe flow.

The transition between laminar flow and turbulent flow occurs in a zone that is basically unstable. For engineering purposes no account is taken of this zone and the critical Reynolds number at which transition will occur is arbitrarily set at 2300. This gives two relations:

(4.5) $Re < 2300.$ $= 64./Re$

$Re > 2300.$ equation (4.4)

The resistance factor is non-dimensional.

To arrive at the loss coefficient C_i needed for the system solution the definition of given by Schlichting [Ref.1] is used.

(4.6a) $$\frac{P_1 - P_2}{L} = \frac{\lambda}{D} \frac{\rho}{2} U^2$$

From this definition the loss coefficient becomes :

(4.6) $$C = \lambda \frac{L}{D} \frac{\rho}{2A^2}$$

λ by equation (4.4) or (4.5)

L length of pipe

D diameter of pipe

ρ specific density

A cross sectional area of pipe

In equations (4.4) and (4.3) the Reynolds number is used. The Reynolds number for pipes is defined as:

$$\text{(4.7)} \qquad Re = \frac{U\,D}{\nu} = \frac{Q\,D}{A\,\nu}$$

The value of the coefficient for roughness k -or as is used more often e - depends obviously on the particular pipe. It must be remembered that this is the value for the "equivalent sand roughness" and not necessarily a function of say the surface finish.

> The combination of analytical and empirical data contained in equation (4.3) and (4.4) are based on the artificial roughening of the surface by applying regular shaped sand grains.

It is therefore usual in commercial application to describe given types of pipes with some equivalent sand roughness. Data from two sources is included in Table 4.1.

IV.- 2.2 Conduits of non-round cross section.

Schlichting [Ref.1] gives a summary of the experimental work conducted on various cross sections. It is obvious that it is not possible to cover all conceivable shapes in a general manner. However, in conduits of uniform cross section (shape and area) the concept of hydraulic radius has proven successful at least in the turbulent flow area. The loss coefficient for such a conduit is defined by:

$$C = \frac{L}{D_H} \frac{\rho}{2A^2} \lambda \tag{4.8}$$

$$D_H = \frac{4 \times \text{Area}}{\text{wetted perimeter}}$$

All other symbols are as defined for equation (4.5)

The values for are computed using equation (4.4) or (4.6) with the Reynolds number defined by

$$Re = \frac{D_e}{\nu} \frac{Q}{A} \tag{4.9}$$

D_e equivalent diameter of cross sectional area.

In the laminar zone discrepancies between resistances calculated with equation (4.8) and resistances experimentally determined has been observed [Ref. 1 p. 576].

Serious discrepancies occur also in conduits where the shape of the conduit changes (Area remains constant) [Ref. 1, p.518]. For the analysis used here, equations (4.8) and (4.9) will be applied.

IV.- 2.3 Changing cross section

Two cases must be distinguished, one where the cross section reduces and the second where the cross section increases.

Area reduces:

The obvious extremes of this type of flow are a sudden contraction (L = 0) on one hand and no reduction on the other.

Some of the early work in terms of loss coefficients goes back to Weisbach [Ref. 9] .

He assumed that all the losses, are essentially occurring at the end of the reducer and determined loss coefficients as a function of geometry.

More recent Gardel [Ref. 10] reported on an investigation conducted at the Ecole Polytechnique in Lausanne (Suisse).

Again these results are correlated on a purely empirical basis, with only geometric parameters entering into the relation.

(Geometry includes installation effects).

From the description given by Gardel [Ref. 10] the tests were conducted in the Reynolds number range from .6 x 10^6 to 1.0 x 10^6. While a certain amount of Reynolds number dependence should be expected, these test results are being acce ted as typical .

(4.10) $$\Delta h = \left[\frac{1.03 - 0.03\ b}{1.-(1.-a)(1.032\ b + 1.38\ a^{1.48} b^{0.7})(1.495 - b^{.49})} - (c+f) \right]^2$$

S_o — area of throat or smallest cross section of reduction

S_1 — area upstream of the entrance to the reducer

S_2 — area downstream of the reducer.

$a = \frac{S_o}{S_1}$ — factor recognizing the amount of reduction in area

$b = \frac{\alpha}{360}$ — where α is the total included angle of the reducer

Note: Gardel's tests were done with α varying from almost zero through 180 degrees (orifice) through 360 degrees (diffusing inlet).

$c = \frac{S_o}{S_1}$ factor recognizing downstream conditions

f factor of convergence(divergence)

$b < 0.6 \qquad f = 0$

$b < 0.8 \qquad f = (1. - c)(b - 0.6)^2$

$b > 0.8 \qquad f = (1. - c)[(b - 0.6)^2 + 525.(b - 0.8)^4]$

$\Delta h = \frac{\Delta H}{V_o^2/2g}$ dimensionless head loss.

To change the value of h to the definition of CL the following modification is performed :

$$C = \frac{\rho}{2}\left(\frac{1}{A_2^2} - \frac{1}{A_1^2} + \frac{\Delta h}{A_2^2}\right) \qquad (4.11)$$

A_1 entrance area

A_2 exit area

In order to solve equation (4.10) the angle α is required. This is determined from :

$$tg\ \alpha = \frac{D_1 - D_2}{L} \qquad (4.12)$$

In case of flow reversal, the reducer obviously becomes a diffuser which will be treated next.

Area increases

Recent work in the field has indicated the strong dependence of losses within the diffuser on the incoming Boundary-layer (both shape and distribution).

A complete survey of recent literature is contained in [Ref. 1], and will not be reported here.

Values of a global loss-coefficient can not readily be extracted from these reports, mainly due to the depence mentioned above.

Gardels work [Ref.10] mentioned earlier, covers the full range of b.

With the definition of b given earlier, any value of b in excess of 0.5 represents a diffuser(area increase).

Thus equations (4.10) and (4.11) given earlier are applicable.However equation (4.12) is modified as follows :

(4.12a) $$tg\, b = \frac{D_2 - D_1}{L} \qquad \alpha = 360. - b$$

IV.- 2.4 Pipe Bends

The losses associated with pipe bends were extensively investigated by [to [Ref.11,12].who also succeeded in correlating experimental data published previously.

> His careful measurements indicate that in addition to a loss within the bend itself, an additional loss occurs downstream of the bend. The data is correlated on the basis of a non-dimensional factor of Reynolds number and nondimensional bend rad us

Since interest is in the total loss due to the bend the total loss coefficient as defined by Ito will be used.

(4.13) $$k = 0.00241\ \alpha\ \theta\ Re^{-.17}\ (R/r)^{.84}$$

θ included angle of the bend

R radius of bend

r pipe radius

Re flow Reynolds number based on diameter.

The factor α is defined as:

(4.14a) $\theta = 45°$ $\quad \alpha = 1. + 14.2(R/r)^{-1.47}$

(4.14b) $\theta = 90°$

$(R/r) < 19.7 \quad \alpha = 0.95 + 17.2(R/r)^{-1.96}$

$(R/r) > 19.7 \quad \alpha = 1.0$

(4.14c) $\quad \theta = 180° \quad \alpha = 1. + 116(R/r)^{-4.52}$

For θ between the values given, interpolation or where necessary extrapolation will be used.

The definition of k reqires again an adjustment in terms of the loss coefficient.

(4.15) $$C = k \frac{\rho}{2} \frac{1}{A^2}$$

As noted earlier, commercial fittings will be treated as nodes later on.

Equation (4.13) is valid for values of $Re(r/R)^2 > 91$. Any designated bend with a value less than the above will be treated as a fitting.

IV.- 3. NODES

IV.- 3.1 Reservoirs

It is very difficult to describe reservoir intakes in a general manner, for this reason a number of assumptions will have to be made.

a) Large reservoir:

These types occur in hydro-power installations, water works, etc. These structures are usually equipped with cleaning racks, slots for shut-off devices, etc. In all of these cases the head loss will be a function of the configuration. For this reason ,data for specific cases should be supplied for the problem under consideration. However, to permit a general aproximation of the expected losses it is assumed that the design of the intake is done in such a way that it results in the smallest possible head-loss.

Flow: Reservoir to conduit

$$C = \frac{\rho}{2A^2}(1. + .0001) \tag{4.16}$$

Flow: Conduit to reservoir

$$C = 0 \tag{4.17}$$

The data in equation (4.16) are based on [Ref. 9], equation (4.17) is developed from the momentum, energy and continuity equations.

b) Pipe Intakes:

These occur usually in installation where normal pipe fittings are used. It is not unusual to weld a standard pipe to a large tank and use this as an intake. It is expected that not more than normal industrial care is exercised in such an installation. These assumptions give .

Flow: Tank into conduit

(4.18) $$C = \frac{\rho}{2A^2} (1. + .15)$$

Flow: Conduit to tank

(4.19) $$C = 0$$

References mentioned earlier were used.

In all of the equations, A denotes the conduit area.

IV.- 3.2 PIPE CONNECTIONS

The data developed by Gardel [Ref. 10] will be used. This data was mentioned earlier in connection with gradually changing cross sections.

To be consistent equation (4.10) is re-numbered and re-written:

(4.20) $$\Delta h = \left[\frac{1.03 - .03\ b}{1.-(1.-\ a)(1.032\ b + 1.38\ a^{1.48}\ b^{0.7})(1.495 - b^{.49})} - (c+f) \right]^2$$

(4.21) $$C = \frac{\rho}{2} \left(\frac{1}{A_2^2} - \frac{1}{A_1^2} + \frac{\Delta h}{A_0^2} \right)$$

Since many configurations are now possible Figure 5 shows the definition used for a large number of cases.

The quantities used in equation (4.20) and (4.21) are defined as follows. (see Figure 5):

$A_1 = \frac{\pi}{4} D_1^2$ upstream conduit area

$A_2 = — D_2^2$ downstream conduit area

A_0 area of smallest cross section For our purpose this is either A_1 or A_2

$a = \frac{A_0}{A_1}$ upstream effect.

$c = \frac{A_o}{A_2}$ downstream effect.

$b = \frac{B^o}{360^o}$ non-dimensional included cone angle.

f additional downstream effect

b <.6 f = 0

b <.8 $f = (1.- c)(b - .6)^2$

b >.8 $f = (1.- c)[(b -.6)^2 + 525.(b -.8)^4]$

IV.- 3.3 PIPE ELBOWS

As in the case of pipe bends it is intended to use the data developed in [Ref.12]

Mitered elbows are usually designed with guide vanes making them equivalent to large radius elbows (R/r = 4 to 6). Thus a large variety of types can be covered with the simple expediency of changing the value of R/r. Because of the wide range of (R/r) expected in the actual applications the full data developed by Ito [Ref. 12]Pwill be used.

(4.22) $k = .00873\ \alpha\ f\ \theta\ (R/r)$ $Re(r/R)^2 < 91$

(4.23) $k = .00241\ \alpha\ \theta\ Re^{-.17}(R/r)^{.84}$ $Re(r/R)^2 > 91$

θ angle of bend

R	radius of bend
r	inside radius of elbow cross section
Re	Reynolds number based on diameter of elbow
f	friction factor incurred in curved pipes given by [Ref.11].

$$\frac{\lambda}{\lambda_o} = [Re\ (\frac{R}{r})^2]^{.05}$$

where λ_o is the friction factor for a straight pipe of equal dimensions

$\theta = 45°$	$\alpha = 1. + 14.2(R/r)^{-1.47}$
$\theta = 90°$	
$(R/r)<19.7$	$\alpha = .95 + 17.2(R/r)^{-1.96}$
$(R/r)>19.7$	$\alpha = 1.$
$\theta = 180°$	$\alpha = 1. + 116.(R/r)^{-4.52}$

To obtain the loss coefficient defined earlier:

$$C = k\ \frac{\rho}{2A^2} \tag{4.24}$$

IV.- 3.4 Y-Branches (Tees)

The most recent work in this field is due to Gardel [13]. Before Gardel, Thoma of Munich (1926-1931) and Favre of

Zurich (1937) established experimental and analytical data on the subject. Gardel does take the experimental work by Thoma into consideration in developing purely experimental correlations.

> While no correlation with Reynolds number has been attempted the model data was obtained at sufficiently high Reynolds number to assure turbulent flow in the pipe lines.

Figure 6 shows in detail the configuration used during the tests and the definitions given by Gardel.

> The head losses defined by Gardel are the ones appearing in the hydraulic gradients at the Y-branches. The hydraulic gradients are determined from upstream and downstream measurements. This implies that the data is applicable to Y-branches in continuous pipe lines only.

The data is presented in relative values which are defined as follows:

Relative flows

$Q_\beta > 0$ $\qquad q_\alpha = 1.$ $\qquad q_\beta = \frac{Q_\beta}{Q_\alpha}$ $\qquad q_\gamma = \frac{Q_\gamma}{Q_\alpha} = 1.- q_\beta$

$0 < q_\beta < 1$

$q_\beta < 0$ $\qquad q_\alpha = \frac{Q_\alpha}{Q_\gamma} = 1. + q_\beta \quad q_\beta = \frac{Q_\beta}{Q_\gamma} \quad q_\gamma = 1.$

$-1 < q_\beta < 0$

Relative head losses: (loss in total energy)

$$h_\beta = 2g\,\frac{H_\beta - H_\alpha}{V_r^2} \qquad h_\gamma = 2g\,\frac{H_\gamma - H_\alpha}{V_r^2} \qquad h_{\gamma\beta} = 2g\,\frac{H_\gamma - H_\beta}{V_r^2}$$

where V_r is the reference flow used in defining q_β.

In addition the following is defined (refer to Figure 6).

$\varphi = \left(\frac{D_\beta}{D_\alpha}\right)^2$ $\qquad D_\alpha = D_\gamma$

δ as defined in Figure 6.

$\rho = 2\,\frac{r}{D_\alpha}$ radius of edge as defined in Figure 6.

With these definitions Gardel was able to correlate his, as well as earlier data as follows:

(4.25) $$_{\pm}h_\psi = {}_0h_\psi(1 - |q_\beta|)^2 + {}_{\pm 1}h_\psi q_\beta^2 + {}_{\pm}A_\psi q\ (1 - |q_\beta|)$$

where ψ takes on the values β, γ or $\gamma\beta$.
The leading subscript (i.e. $_{\pm}h$) refers to the sign of q_β.

For all cases the following applies :

$_{o}h_{\beta} = - .95$

$_{o}h_{\gamma} = - .03$

$_{o}h_{\gamma\beta} = .92$

For $q_{\beta} > 0$

$_{+1}h_{\beta} = - (1.3 \text{ ctg}\frac{\delta}{2} - .3 + \frac{.4 - .1\varphi}{\varphi^2})(1 - .9\sqrt{\rho/\varphi})$

$_{+1}h_{\gamma} = -.35$

$_{+1}h_{\gamma\beta} = {}_{+1}h_{\gamma} - {}_{+1}h_{\beta}$

$_{+}A_{\beta} = - 0.4(1. + \frac{1}{\varphi})\text{ctg}\frac{\delta}{2}$

$_{+}A_{\gamma} = 0.2$

$_{+}A_{\gamma\beta} = {}_{+}A_{\gamma} - {}_{+}A_{\beta}$

For $q_{\beta} < 0$

$_{-1}h_{\beta} = 1. + 0.42(\frac{\cos\delta}{\varphi} - 1.) - .8(1. - \frac{1}{\varphi^2})$
$+ (1. - \varphi)(\frac{\cos\delta}{\varphi} - .38)$

$_{-1}h_{\gamma} = 1. + (1.62 - \sqrt{\rho})(\frac{\cos\delta}{\varphi} - 1.) - .38(1. - \varphi)$

$_{-}A_{\beta} = 0$

$_{-}A_{\gamma} = 2. - \varphi$

$_{-}A_{\gamma\beta} = {}_{-}A_{\gamma} - {}_{-}A_{\beta}$

$_{-1}h_{\gamma\beta} = {}_{-1}h_{\gamma} - {}_{-1}h_{\beta}$

To permit an orderly procedure to be followed within the program, the following assumptions are made. :

a) Selection of Branches is based on the normal flow condition expected through the node.

b) The branches into or out of the Y will be treated as beta type branches, since it is expected that a true through flow branch as tested will be the exception rather than the rule.

With these selections, it is possible to calculate the corresponding h . To get to the loss coefficient defined earlier, the following is done.

$$C_\beta = \frac{\rho}{2}\left(\frac{q_\beta^2}{A_\beta^2} - \frac{q_\alpha^2}{A_\alpha^2} + \frac{1}{A^2_{\alpha\, or\, \gamma}} h_\beta\right) \quad (4.26)$$

$$C_{\gamma\beta} = \frac{\rho}{2}\left(\frac{q_\gamma^2}{A_\gamma^2} - \frac{q_\beta^2}{A_\beta^2} + \frac{q_\gamma^2}{A_\gamma^2} h_{\beta\gamma}\right) \quad (4.27)$$

These coefficients must be used in connection with the reference flow (α or γ).

IV.- 3.5 VALVES

The description of Valves now moves into the field of individual designs, and it is not expected that at this time generally applicable performance data can be supplied. The Form of the data to be supplied by the user is described. However as part of this analysis the performance of three types of shut-off and relief valves commonly encountered in hydroelectric installations will be supplied.

a) General Data

The usual manner in which engineering data on valve performance is recorded is of the form of a discharge coefficient .

$$\mu = \frac{Q_{ACT}}{Q_{TH}} \qquad (4.28)$$

Q_{ACT} actual discharge through valve

Q_{TH} $= A_r \sqrt{2g\Delta H}$

A_r valve reference diameter

ΔH is the difference in total energy head as defined by the Bernoulli equation for the particular tests determining μ

$\mu = \mu(s)$ is a function of stroke (i.e. for $S = 0 \ldots \mu = 0$)

If the assumption is made that the valve performance data is based on being installed in a pipe line essentially of the same size as the reference diameter of the valve then equation (4.28) can be re-written:

$$Q_{ACT} = \mu A_r \sqrt{\frac{2}{\rho}(P_1 - P_2) + (V_1^2 - V_2^2)}$$

After re-arranging this gives:

$$C = \frac{\rho}{2}\left(\frac{1}{A_2^2} - \frac{1}{A_1^2} + \frac{1}{\mu^2 A_r^2}\right) \quad (4.29)$$

The value of the discharge coefficient will, for a closed valve, be zero. However, a closed valve negates the basic equation upon which the relation was established in the first place, i.e. in this case the flow in the adjacent pipes is zero and the relation between P and Q across the node can not be defined.

However, in order that the computation of the rest of the system may proceed in the manner indicated, the convention is adopted that whenever mu is zero, the value C becomes as large a number as practicable. This will result in a small flow across the valve being computed, however, this small flow will not influence the reliability of the calculated values in the rest of the system.

To sum up:

i) Data of valve performance must be given as a function of discharge coefficient mu versus non-dimensionalized stroke. The value of mu is for a valve tested in a constant diameter pipe line.

ii) The program will interpolate the performance table and calculate as per equation (4.29).

b) Specific Performances:

The performance values of the following valves are included in the performance-subroutine:

1. Butterfly valves

2. Spherical valves

3. Relief valves (specific design)

The data is shown on Table 4.2 and is based on unpublished general manufacturers specifications.

IV.- 3.6 HYDRAULIC MACHINERY

Just as in the case of valves the actual data used will be a question of application. Design and performance will depend on the individual manufacturer.

The intention of this analysis is again to provide a framework which will accept data in a certain form and use it in the mechanized analysis.

In addition a number of relative performance data based on one manufacturer's design will be included.

i) General data

a) Hydraulic turbines:

Hydraulic turbines performance data is available in the form of a set of two tables:

Table I ϕ vs. Q_{11}) at fixed gate-opening

Table II ϕ vs. HP_{11})

$$\text{unit discharge } Q_{11} = \frac{Q}{D^2 H^{1/2}}$$

$$\text{unit output } HP_{11} = \frac{HP}{D^2 H^{3/2}}$$

$$\text{relative speed } \phi = \frac{D N}{H^{1/2}} \frac{\pi}{60(2g)^{1/2}}$$

Q actual discharge of the turbine for the given gate opening.

HP actual shaft horse-power for the given opening.

N shaft speed in RPM.

D throat diameter of the impeller.

H turbine net head, defined below.

Care must be taken to ensure that all dimensions are in consistent units. i.e. FT-lbs.-sec or m-kg-sec.

Over the years test codes have been developed which define the quantities given above.

Net Head is defined by the ASME test code as the difference between the total energy head at the case intake and the energy head in the tail race canal. (Tail-race canal width is defined for multi-machine installations as the unit spacing).

In the case of European codes, and now the International test code, the upstream energy head is as above, however, the downstream energy head is defined as the pressure head at the draft tube exit increased by the draft tube exit velocity head.

Velocity head in all cases is defined as the discharge of the turbine divided by the relevant cross sectional area.

Since the performance of a complicated structure such as a pump or turbine can as yet not be forecast from theoretical consideration alone, the performance of a model is determined by test and the unit data defined above is then applied at the size and head of the actual machine. The Reynolds number dependency of the unit data is known to exist and will be taken into consideration in every application. However, the definitions given above are still valid, the numerical value of the unit data only is changed.

> However, since there are as as many "step up" relations as there are major turbine manufacturers and consulting engineers, it is to be assumed for this particular purpose that the unit performance values given, are for the actual installation. Furthermore it is also to be assumed that the definition of Net Head given by the ASME test code is applicable.

Starting with the definition of Q and Net-head, the following is rewritten:

$$P_1 - \frac{\rho}{2}\left(\frac{2g}{Q_{11}^2\, D^4} - \frac{1}{A_1^2}\right) Q^2 = \rho g S \quad (4.30)$$

P_1 pressure in inlet section

A_1 area of inlet section

S submergence of turbine referred to . center-line of the intake

From the above the following is obtained:

(4.31) $$C = \frac{\rho}{2}\left(\frac{2g}{Q_{11}^2 D^4} - \frac{1}{A_1^2}\right)$$

If the turbine is installed with a long conduit following it, then equation (4.30) must be rewritten.

(4.32) $$P_1 - P_2 - \frac{\rho}{2}\left(\frac{1}{A_2^2} - \frac{1}{A_1^2} + \frac{2g}{Q_{11}^2 D^4}\right) = 0$$

For this case C becomes

(4.33) $$C = \frac{\rho}{2}\left(\frac{2g}{Q_{11}^2 D^4} - \frac{1}{A_1^2} + \frac{1}{A_2^2}\right)$$

Here again the convention is adopted that whenever Q is zero C will be set equal to a very large number, so that the calculation procedure for the rest of the system can proceed without alteration.

b) Pumps:

Engineering data on pumps is usually presented in one of two ways:

I. Q-H-relations at a fixed speed .

Data in this form does not require any special manipulation, it is directly interpolated for the specific conditions.

II. specific data:

$$\varphi = \frac{Q}{N\ D^3} \qquad \psi = \frac{gH}{N^2\ D^2} \qquad k = \frac{HP}{\rho N^3 D^5}$$

D diameter of runner tip

The head in this case is the net energy head between pump inlet and outlet.

Rearranging and combining the above equation together with the definition of H gives:

(4.34) $$P_1 - P_2 - \frac{\rho}{2}\left(\frac{1}{A_2^2} - \frac{1}{A_1^2} + 2\frac{\psi}{\varphi^2 D^4}\right)Q^2 = 0$$

From the above the following is obtained:

(4.35) $$C = \frac{\rho}{2}\left(\frac{1}{A_2^2} - \frac{1}{A_1^2} + 2\frac{\psi}{\varphi^2 D^4}\right)$$

In the case of a pump, unless the cause of $\varphi = 0$ is the closing of wicket gates (pump-turbines), equation (4.33) is valid and no replacement of C_i by a large number is indicated. (Other than to protect against aborting by the computer).

ii) Internal Supplied Data:

Only one typical Francis turbine of specific speed of 40 is described in the program. The description shown on Table 4.2 is in terms of values relative to peak performance.

The actual values of the peak performance must be given as Q, HP, N, D, and head for the peak performance point, as well as the wicket gate opening for which the flow conditions are to be calculated.

The internal subroutines then substitute the actual values and submit those for calculation procedure.

For the moment, no internal data on pumps is supplied.

V.- COMPUTER PROGRAM

A full and detailed description of the computer program developed will not be given here, only a general description as well as some results arrived at through the program will follow.

Figure 7 shows a MACRO-flow chart of the program.

There are five distinct phases to the execution:

A) Data Input

B) Checking of Input Data

C) Printing of Input Data

D) Initializing all constants, etc.

E) Iterative solution and printing of results.

Due to the size of the program (mainly caused by the data storage) the program must be run in segments.

This segmentation is done according to the above divisions, the control of running the segments is handled by one MAIN program.

Despite the segmentation, the programs requirements for memory are 48 k of 36-bit words. This can possibly be reduced if data storage in quick access discs in lieu of core residency is available, but no attempt was made at this point to economize or better optimize the running of the program.

V.- 1 Data Input

The Data Input segment accepts all data with "Namelist"-format, a format that is essentially free field, with only minor restrictions as to Format.

The following data is required by the program.

1) General Information

   Data such as the characteristics of the fluid are given here;

2) Description of Conduits

3) General Description of Nodes

4) Physical Description of Nodes

5) Performance Characteristics of Nodes.

V.- 2 Checking of Data

A number of checks are performed on the data to guard against possible fatal errors due to faults in the data input.

i) All index numbers are checked for duplications.

ii) Verify that all descriptions required by the nodes are available. Where internal data was requested , this data is supplied.

iii) The existence of all nodes requested by the conduits is checked, at the same time the conduits connected to a given node, are recorded for the node.

iv) Check if each node has the required pipes identified.

v) Check if there is an energy source present which will drive the fluid through the conduits.

This part of the program will, if too many errors are recognized, set a switch to terminate computations after the next step (Printing).

V.- 3 Printing of Input Data

The print out is arranged to be self-explanatory. It is intended to record in great detail the input data as the computer has seen it.

This segment can be by-passed if desired.

As mentioned earlier, if errors have occurred the program will terminate after completion of the print out.

V.- 4 Initializing

As the name indicates, constants and other data such as specific performance curves are set-up so that the next segment can proceed with the least amount of repetitive operations.

V.- 5 Iterative Solution

This segment contains essentially the following 4 elements:

a) Set up the matrix coefficients for use in the solution (linearized)

b) Solve the system of equations

c) Check if solution has been obtained

if "yes" go to step (d)

if "no" start again at (a)

d) Print results

VI TEST EXAMPLES AND DISCUSSION

The computer program has as yet not been tested to its fullest capacity. During the development of the program a relatively small (six pipes) system was used to check the various program options and built-in data. The print-outs for three specific conditions of this small system are enclosed as Appendix A.

The basic outline of the Network (pipe line) is shown in Fig.8. The differences between the examples are as follows.

a) Example 1

The two valves (Node Nr. 4 and 6) are both fully open. The resulting flow and pressure condition must be symmetrical for the two branches.

b) Example 2

Valve Nr.4 is half closed. Losses at the two branches of the Y should be of comparable magnitude.

c) Example 3

The description of conduits 2 and 3 has the node connections reversed. Otherwise the conditions are identical with Example 2. This will result in a negative flow in these conduits.

The results of Examples 1 and 2 are shown in Fig.9. Example 3 is- with the exception of the minus signs on flows in conduits 2 and 3- a duplicate of Example 2.

In comparing different methods using computers the question of computer time invariably appears. This time or cost is usually represented by the number of occasions the computer repeats given steps (number of iterations).

It is usually very difficult to draw valid conclusions from the data obtained on one computer, unless the various methods are compared on the same machine.

Nevertheless a few remarks concerning the method developed in ths report are offered.

The number of iterations necessary to arrive at the reults shown was approximately 7.

One iteration in this case includes a complete setting up of the system matrix, including linearization, and a full solution by a modified Gaussian reduction method.

A solution within 1% of the final solution has been attained within 4 iterations for the system shown.

All of the above solutions were obtained starting from an initial "Null"-vector. Obviously the number of iterations can be reduced if a better estimate of the final solution is available.

While the number of iterations will undoubtedly increase with the number of pipes to be analysed, there is some evidence existing that this will not be a direct relationship.

During the development of the program, the change from a straight through pipe line (conduits 5 and

6 were not included) to the arrangement shown in Fig.8 did not result in any change in the number of iterations required to find a solution.

From the limited use the program has had sofar, it appears that the inclusion of socalled minor losses into the computation of flow through pipe-networks has been successfully accomplished.
It is hoped that the program will permit a rational study of minor losses as they are affected by the presence of conduits. Even without any additional experimental work ,the author believes that analysis of published data on observed pipe-network behaviour could contribute significiantly to the problem at hand.

REFERENCES

[1] H. Schlichting Boundary layer theory
6th Edition McGraw-Hill 1968

[2] A. Ralston First course in numerical analysis
McGraw-Hill 1965

[3] L. Prandtl Neuere Ergebnisse der Turbulenz-forschung
Z. VDI 77,pp.105 (1933)

[4] H. Blasius Das Aehnlichkeitsgesetz bei Reibungsvorgaengen in Fluessigkeiten
Forschg.Arch.& Ing.Wesen Nr 131 (1913)

[5] Th.von Karman Mechanische Aehnlichkeit & Turbulenze
Nach.Ges.Wiss. Goettingen,
Math.Phys.Klasse 58 (1930)

[6] J. Nikuradse Stroemungsgesetze in rauhen Rohren
Forschg.Arch.& Ing,Wesen Nr. 361 (1933)

[7] L.F. Moody Friction factors for Pipe flow
Trans.ASME 66, 671 (1944)

[8] C.F. Colebrook Turbulent flow in pipes with particular reference to the transition region between the smooth and rough pipe laws.
Jour.Inst.Civ.Eng. (1939)

[9] A. Weisbach Die Experimental Hydraulic
Engelhardt,Freiburg, (1855)

[10] A. Gardel Perte de charge dans un Etrangement Conique
Bulletin Technique de la Suisse Romande Oct. 1962

[11] H. Ito Friction factors for turbulent flow in curved Pipes
ASME Paper 58-SA-14,Detroit 1958

[12] H. Ito Pressure loss in smooth Pipe bends
ASME Paper 59-HYD 4, Ann Arbour,1959

[13] A. Gardel Les Pertes de charge dans les Ecoulement au travers de Branchement en Te.
Bulletin Technique de la Suisse Romande
Vol.33,Nr.9 & 10, 1957

[14] H. Cross — Analysis of flow in Networks of Conduits or Conductors. Bull.Univ.of Illinois Eng.Exper.Station, 1934

[15] D. Frederick, T.S.Chang — Continuum Mechanics. Allyn and Bacon,Boston 1969

[16] V.C. Streeter, E.B. Wylie — Hydraulic Transients. McGraw-Hill book Co. 1967

[17] C.F. Lam, M.L. Wolla — Computer Analysis of Water Distribution Systems. Part I -Formulation. Proc.ASCE, Jour.Hyd.Eng.pp335, 1972

[18] H.K. Kesaran, M.Chandrashekar — Graph-Theoretic Models for Pipe Network analysis. Proc.ASCE,Jour.Hyd.Eng. pp.345 ,1972

[19] H. Mueller, J. Stratman — Friction Losses in Penstocks. Sulzer-Technical Review 3/1964

Description	k in inches [Ref.7]	[Ref.19]
Glass, brass, copper, lead	smooth	-
drawn tubes and pipes	.00006	-
Structural and forged steel (welded pipes)		
New	.0018	.0018 - .004
New, asphalted(bituminized)	.0047	.0005 - .0008
Bitumen, partly flaked off	-	.003 - .004
slighly encrusted	-	.004 - 008
medium encrustation	-	.02 - .04
advanced encrustation	-	.4 - 1.2
cleaned after long use	-	.004 -.008
galvanized	.006	-
Cast Iron	.01	-
Wood	.007 -.03	-
Concrete	.012 -.12	-
Rivetted steel pipes	.035 -.35	-

EQUIVALENT SAND ROUGHNESS

Table 4.1

Relative stroke	Discharge coefficient: Butterfly valve	Spherical valve	Relief valve
.0	.0	.0	.0
.2	.045	.10	.27
.4	.112	.21	.50
.6	.254	.41	.67
.8	.59	.70	.80
1.0	.90	1.0	.85

VALVE PERFORMANCE

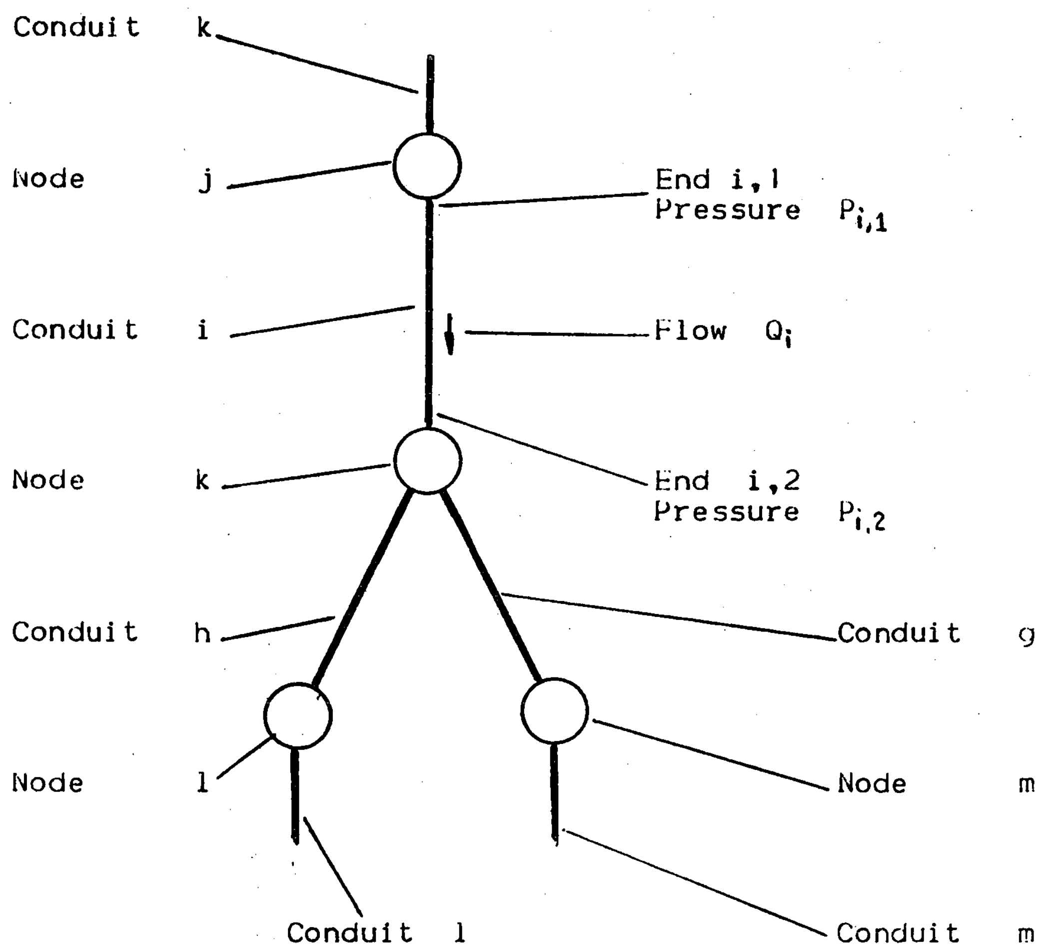
Table 4.2

WG	Relative Values O	HP	Q
.438	.890	.355	.418
	1.000	.372	.450
	1.092	.307	.422
	1.42	.00	.315
.625	.753	.605	.683
	.890	.621	.673
	1.0	.698	.646
	1.092	.573	.630
	1.23	.419	.578
	1.52	.00	.468
.812	.753	.788	.886
	.890	.828	.855
	1.0	.806	.829
	1.092	.755	.795
	1.23	.640	.756
.938	.753	.880	.995
	.890	.950	.960
	1.00	.940	.943
	1.092	.875	.908
	1.23	.738	.855
	1.605	.00	.632
1.062	.753	.968	1.088
	.890	1.04	1.062
	1.0	1.035	1.035
	1.092	.985	1.005
	1.23	.840	.958

WG	Relative Values O	HP	Q
1.19	.753	1.050	1.185
	.890	1.112	1,12
	1.0	1.112	1.12
	1.092	1.085	1.095
	1.23	.906	1.035
1.312	.767	1.13	1.27
	.893	1.178	1.23
	1.0	1.178	1.205
	1.11	1.135	1.175
	1.23	1.00	1.135
	1.65	.00	.78
1.43	.788	1.192	1.34
	.895	1.232	1.305
	1.02	1.232	1.28
	1.10	1.192	1.265
	1.19	1.11	1.25
1.625	.842	1.27	1.422
	.910	1.285	1.39
	1.00	1.290	1.380
	1.10	1.232	1.335
	1.205	1.065	1.21
	1.68	.00	.885

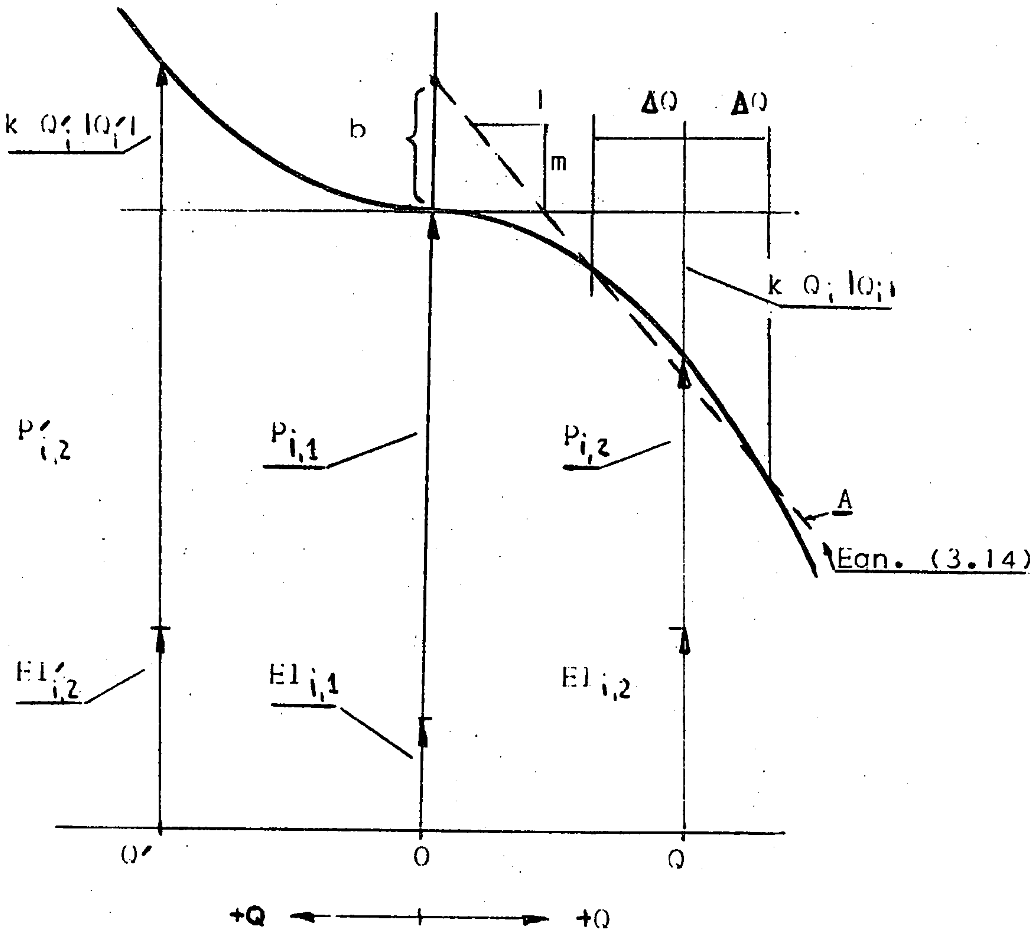
RELATIVE TURBINE PRFORMANCE

Table 4.3



TYPICAL PART OF SYSTEM

Figure 1

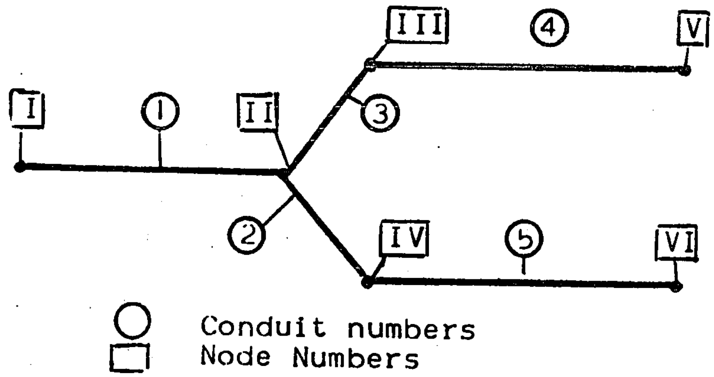


Equation (3.7)

$$P_{i,1} - P_{i,2} - C_i \; Q_i \, |Q_i| - \rho g(El_{i,2} - El_{i,1}) = 0$$

EQUATION (3.7)

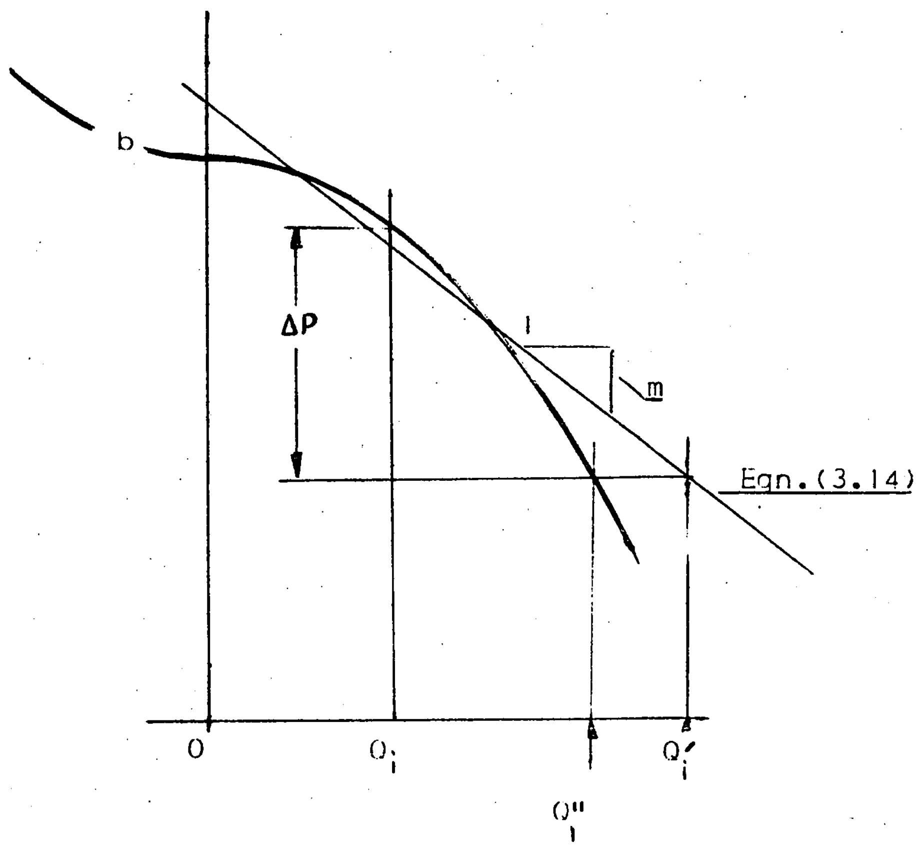
Figure 2



$$
\begin{array}{c}
1\\ 2\\ 3\\ 4\\ 5\\ I\\ II\\ \\ \\ III\\ \\ IV\\ \\ V\\ VI
\end{array}
\begin{bmatrix}
1 & -1 & m & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0\\
0 & 0 & 0 & 1 & -1 & m & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0\\
0 & 0 & 0 & 0 & 0 & 0 & 1 & -1 & m & 0 & 0 & 0 & 0 & 0 & 0\\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 & -1 & m & 0 & 0 & 0\\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 & -1 & m\\
1 & 0 & m & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0\\
0 & 1 & 0 & 0 & 0 & 0 & -1 & 0 & m & 0 & 0 & 0 & 0 & 0 & 0\\
0 & 1 & 0 & -1 & 0 & m & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0\\
0 & 0 & 1 & 0 & 0 & -1 & 0 & 0 & -1 & 0 & 0 & 0 & 0 & 0 & 0\\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & -1 & m & 0 & 0 & 0\\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & -1 & 0 & 0 & 0\\
0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & -1 & 0 & m\\
0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & -1\\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 & m & 0 & 0 & 0\\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 & m
\end{bmatrix}
\begin{bmatrix}
P_{1,1}\\ P_{1,2}\\ Q_1\\ P_{2,1}\\ P_{2,2}\\ Q_2\\ P_{3,1}\\ P_{3,2}\\ Q_3\\ P_{4,1}\\ P_{4,2}\\ Q_4\\ P_{5,1}\\ P_{5,2}\\ Q_5
\end{bmatrix}
=
\begin{bmatrix}
B_1\\ B_2\\ B_3\\ B_4\\ B_5\\ B_{I}\\ B_{II}\\ B_{II}\\ 0\\ B_{III}\\ 0\\ B_{IV}\\ 0\\ B_{V}\\ B_{VI}
\end{bmatrix}
$$

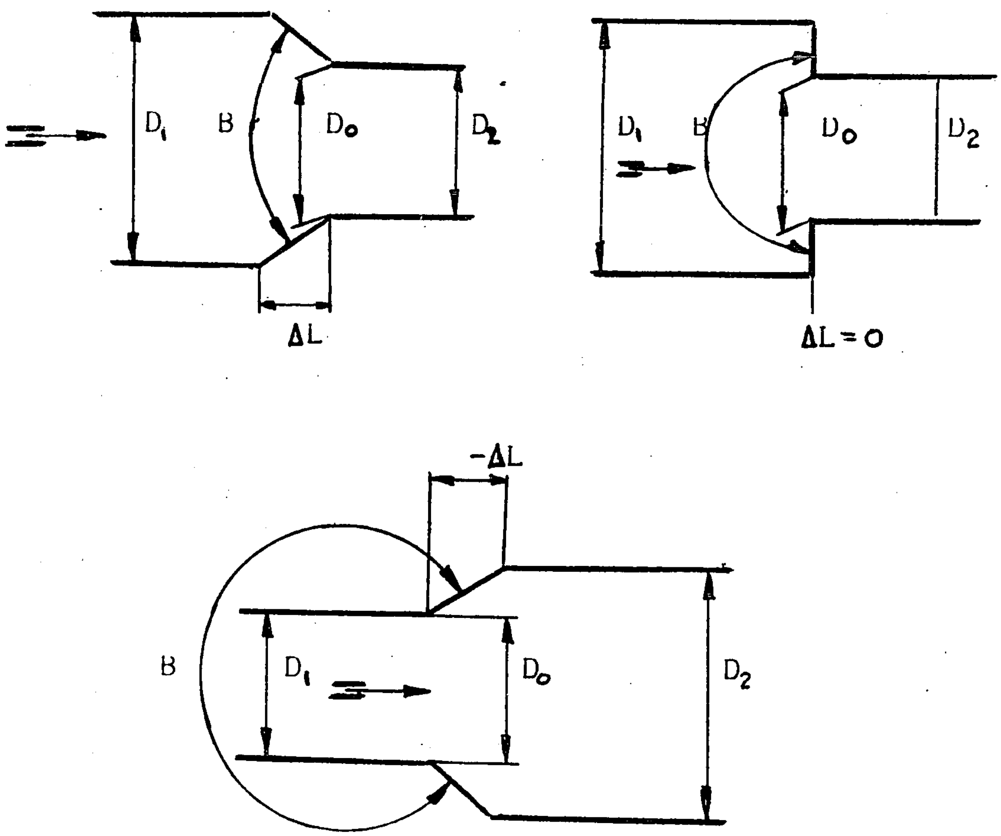
SYSTEM and SYSTEM MATRIX

Figure 3



DETERMINATION OF Q"

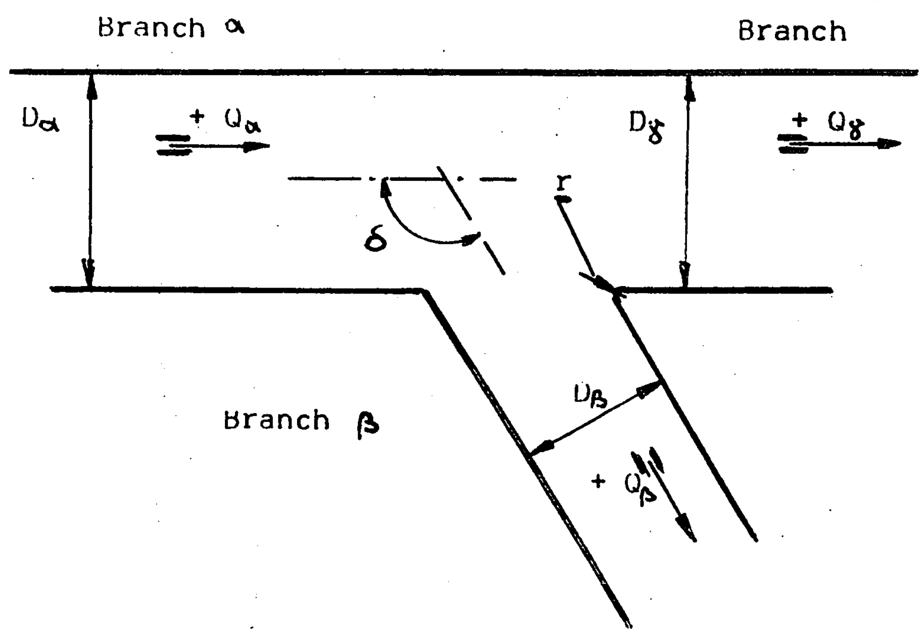
Figure 4



Arrow denotes direction of flow

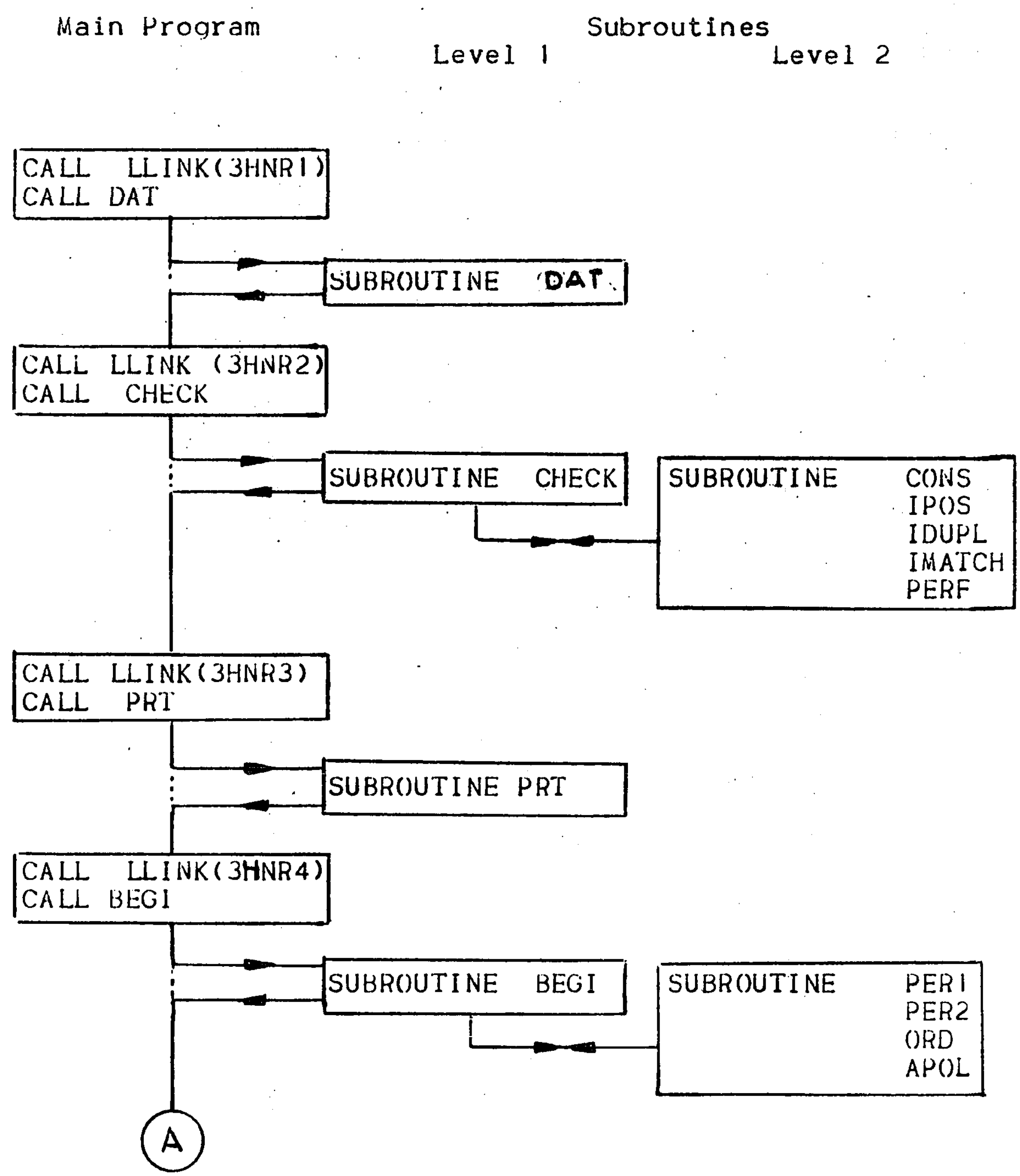
TYPICAL CONNECTIONS

Figure 5



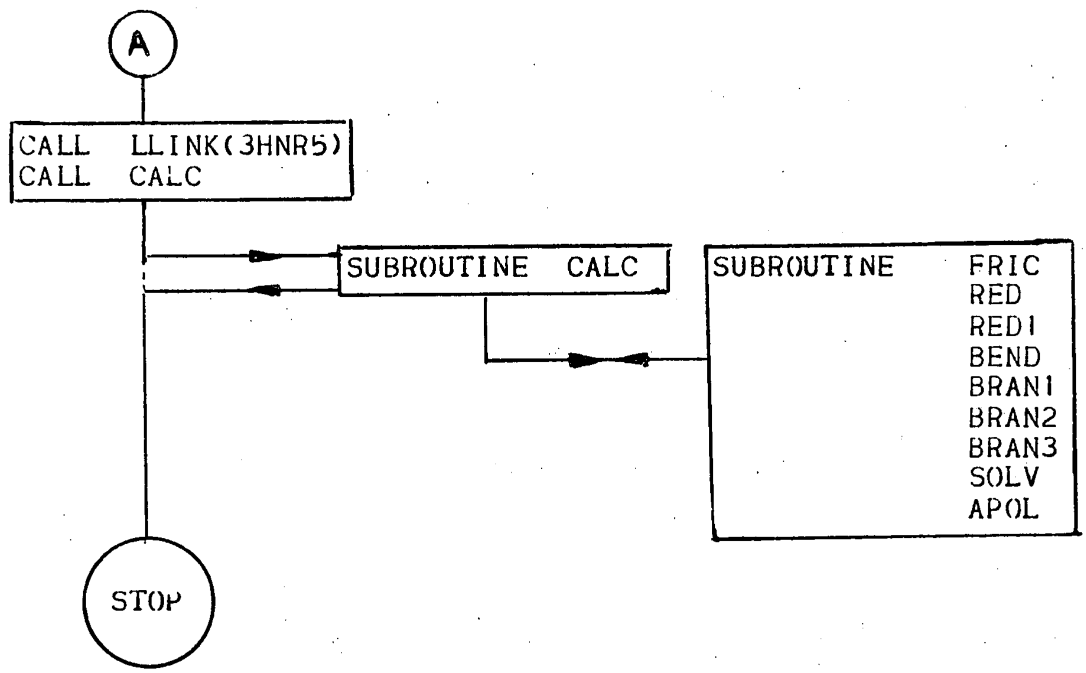
DEFINITIONS for Y - BRANCHES

Figure 6



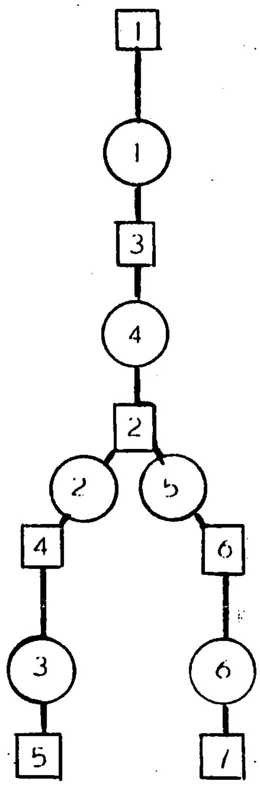
MACRO - FLOW CHART

Figure 7



MACRO - FLOW CHART

Figure 7 a



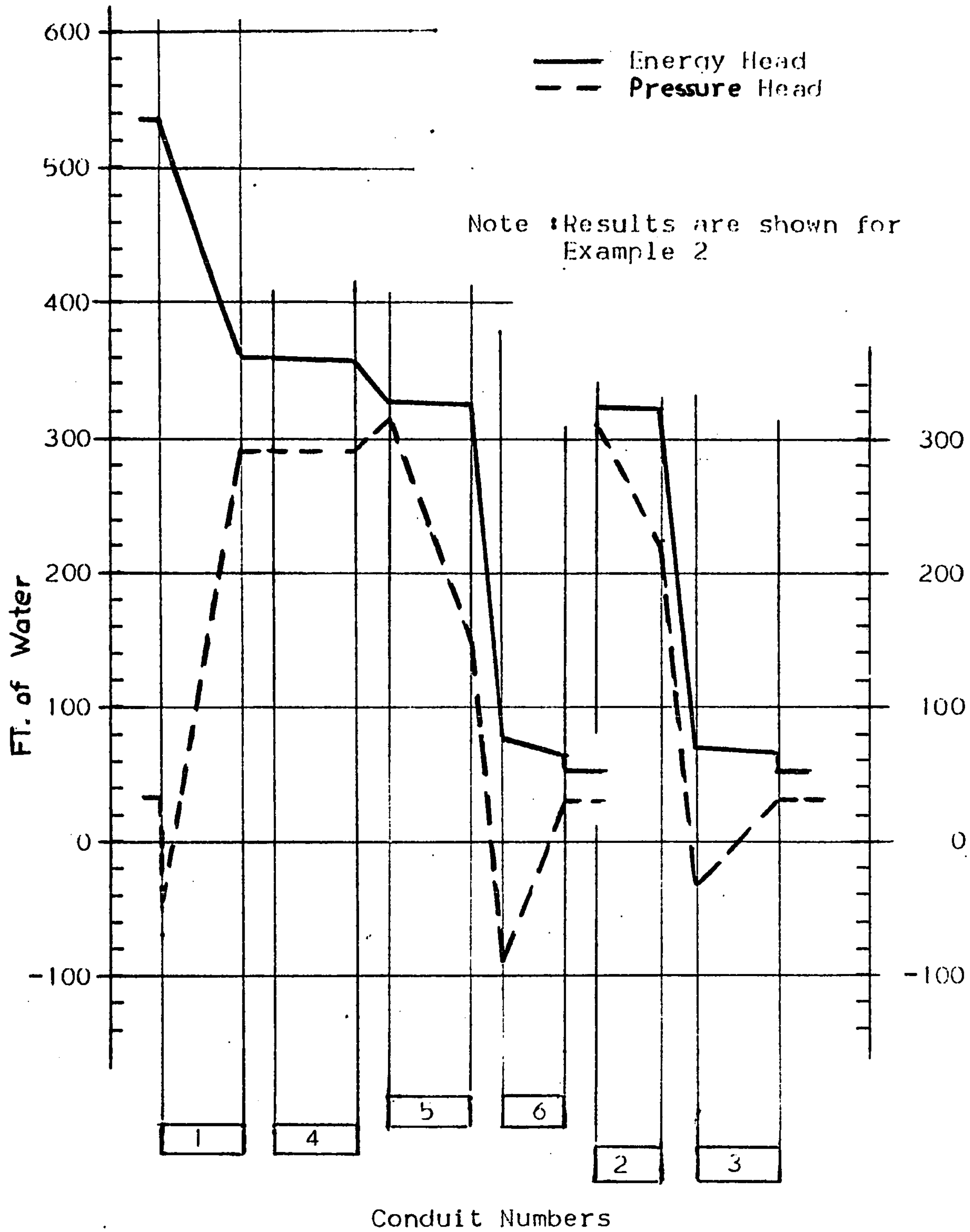
Reservoir	Elevation 500. Submergence 30.
Conduit	constant diameter
Pipe connect.	equal diameter
Conduit	Pipe bend
Y-Branch	Elevation 0.
Conduits	Reducers
Valves	Elevation 0.

Note : Valve 4 is half closed for Example 2

Conduit	Diffusers
Reservoir	Elevation 20. Submergence 30.

SCHEMATIC ARRANGEMENT OF EXAMPLES

Figure 8



Results of Calculation
Figure 9

APPENDIX A

```
10$        EXECUTE
20$        LIMITS  20,50000,,1000
30$        PRMFL   P*,W,S,XK0315T/OUT
40    PROJECT  1177-2,  TESTING
115$        DATA    15
50 $GEN1  IT(1)=1,1,3$
200 $GEN2 AT(1)=32.17,62.4,43.2E+06,10.0,1.2E-05,.1,10.0,.5$
210 $GEN3  LTOT/IT=1,3,5,4,6,7$
220 $PIPE  IT(1)=1,1,1,3,0,I2/AT(1)=5.0,5.0,500.0,
225   -0.005,0.0,0.1,4.32E+09,0.3,500.0,10.0E+09,0.1$
230 $PIPE  IT(1)=2,1,2,4,1,I2/AT(1)=5.0,3.0,20.0,
235   -0.01,10.0,0.2,4.32E+09,0.3$
240 $PIPE  IT(1)=3,1,4,5,0,I2/AT(1)=3.0,5.0,20.0,
245   -0.005,10.,0.05,4.32E+09,0.3$
250 $PIPE  IT(1)=4,3,3,2,0,I2/AT(1)=5.0,10.0,30.0,
255   -0.001,0.0,0.2,4.32E+09,.3,500.0,10.0E+09,0.1$
260 $NODE1  IT(1)=1,1,0,-1,AT(1)=500.0$
265 $NODE1  IT(1)=3,3,0,-1,AT(1)=0.0$
275 $NODE1  IT(1)=4,4,1,5,AT(1)=0.0$
280 $NODE1  IT(1)=5,2,0,-1,AT(1)=20.0$
285 $NODE2  IT(1)=1,1,1,AT(1)=30.,0.1,1.0$
290 $NODE2  IT(1)=3,2,AT(1)=0.,0.,0.$
300 $NODE2  IT(1)=4,6,0,AT(1)=3.$
305 $NODE2  IT(1)=2,1,-1,AT(1)=30.,.1,1.$
310 $NODE3  IT(1)=1,1,I2/AT(1)=0.0,1.0,0.2,0.8,0.4,0.6,
315    0.6,0.4,0.8,0.2,1.0,0.0$
316 $NODE3   IT(1)=2,1,I2/AT(1)=0.5.0.5,0.6,0.4,0.8,0.2,1.0,0.0$
320 $NODE4  IT(1)=5,3,I2/AT(1)=0.0,0.0,0.2,0.33,
325    0.4,0.55,0.6,0.69,0.8,0.77,1.0,0.82$
330 $PIPE  IT(1)=5,1,2,6,1,I2/AT(1)=5.0,3.0,20.0,
335   -0.01,10.0,0.2,4.32E+09,0.3$
340 $PIPE  IT(1)=6,1,6,7,0,I2/AT(1)=3.0,5.0,20.0,
345   -0.005,10.,0.05,4.32E+09,0.3$
350 $NODE1  IT(1)=6,4,1,5,AT(1)=0.0$
355 $NODE1  IT(1)=7,2,0,-1,AT(1)=20.0$
360 $NODE1  IT(1)=2,5,0,-1,4,5,2,AT(1)=0.0$
365 $NODE2 IT(1)=5,3,AT(1)=120.,120.,0.02$
60$        ENDJOB
```

APPENDIX A

PROJECT 1177-2, TESTING EX. 1 PAGE NR. 1

```
O-GENERAL DATA
          DIMENSIONAL SYSTEM USED......FT-LB-SEC
          BASIC DATA
               ACCELER.OF GRAVITY                32.17
               SPEC.WT.OF MEDIUM                 62.40
               MOD.OF COMPR.OF FLUID        0.4320E 08
               VISCOSITY OF FLUID           0.1200E-04
          OTHER INFORMATION
               TRANSIENT CALC.
                     STARTING TIME          0.
                     COMPLETION TIME             10.00
                     RESULTS REQUESTED FOR NODES
                        1        3        5        4

                        6        7
```

APPENDIX A

PROJECT 1177-2, TESTING EX. 1

I-CONDUIT DESCRIPTION

C O N D U I T

NR.	TYPE		DESCRIPTION	
1	ROUND SECTION		NODE AT END 1	1
			END 2	3
			ANCHORED THROUGHOUT	
			I.D. END 1	5.000
			I.D. END 2	5.000
			LENGTH	500.0
			WALL ROUGHNESS	0.5000E-02
		WALL 1	THICKNESS	0.1000
			MOD.OF ELAST.	0.4320E 10
			POISSON RATIO	0.3000
		WALL 2	THICKNESS	500.0
			MOD.OF ELAST.	0.1000E 11
			POISSON RATIO	0.1000
2	ROUND SECTION		NODE AT END 1	2
			END 2	4
			ANCHORED ONE END ONLY	
			I.D. END 1	5.000
			I.D. END 2	3.000
			LENGTH	20.00
			WALL ROUGHNESS	0.1000E-01
		WALL 1	THICKNESS	0.2000
			MOD.OF ELAST.	0.4320E 10
			POISSON RATIO	0.3000
3	ROUND SECTION		NODE AT END 1	4
			END 2	5
			ANCHORED THROUGHOUT	
			I.D. END 1	3.000
			I.D. END 2	5.000
			LENGTH	20.00
			WALL ROUGHNESS	0.5000E-02
		WALL 1	THICKNESS	0.5000E-01
			MOD.OF ELAST.	0.4320E 10
			POISSON RATIO	0.3000

I-CONDUIT DESCRIPTION (CONTD.)

CONDUIT NR.	TYPE		DESCRIPTION	
4	PIPE BEND		NODE AT END 1	3
			END 2	2
			ANCHORED THROUGHOUT	
			I.D.OF BEND	5.000
			RAD.OF BEND	10.00
			INCL.ANGLE	30.00
			WALL ROUGHNESS	0.1000E-02
		WALL 1	THICKNESS	0.2000
			MOD.OF ELAST.	0.4320E 10
			POISSON RATIO	0.3000
		WALL 2	THICKNESS	500.0
			MOD.OF ELAST.	0.1000E 11
			POISSON RATIO	0.1000
5	ROUND SECTION		NODE AT END 1	2
			END 2	6
			ANCHORED ONE END ONLY	
			I.D. END 1	5.000
			I.D. END 2	3.000
			LENGTH	20.00
			WALL ROUGHNESS	0.1000E-01
		WALL 1	THICKNESS	0.2000
			MOD.OF ELAST.	0.4320E 10
			5POISSON RATIO	0.3000
6	ROUND SECTION		NODE AT END 1	6
			END 2	7
			ANCHORED THROUGHOUT	
			I.D. END 1	3.000
			I.D. END 2	5.000
			LENGTH	20.00
			WALL ROUGHNESS	0.5000E-02
		WALL 1	THICKNESS	0.5000E-01
			MOD.OF ELAST.	0.4320E 10
			POISSON RATIO	0.3000

II-NODE DESCRIPTION

A-GENERAL DATA

CODES USED

PERFORMANCE NR.	POSITIVE	DATA SUPPLIED
	ZERO	CONSTANTS USED
	NEGATIVE	BUILT-IN DATA USED

NODE NR.	D E S C R I P T I O N N R. DIMENSION	STROKE	PERFORMANCE	REF.ELEVATION OF NODE
1	1	0	-1	500.0
3	3	0	-1	0.
4	4	2	5	0.
5	2	0	-1	20.00
6	4	1	5	0.
7	2	0	-1	20.00
2	5	0	-1	0.

APPENDIX A

PROJECT 1177-2, TESTING EX. 1 PAGE NR. 5

II-NODE DESCRIPTION

B-NODE DIMENSIONS

DESCR. NR.	NODE TYPE	DETAILED DESCRIPTION		
1	RESERVOIR	UPSTREAM		
		SUBMERGENCE		30.00
		LOSS INTO COND.		0.1000
		FROM COND.		1.000
3	PIPE CONN.	ENTR.ANGLE	0.	
		LOSS LG.TO SMALL	0.	
		SMALL TO LG.	0.	
4	VALVE	CENTER		
		SUBMERGENCE	0.	
		NOM.DIA.		3.000
2	RESERVOIR	DOWNSTREAM		
		SUBMERGENCE		30.00
		LOSS INTO COND.		0.1000
		FROM COND.		1.000
5	Y-BRANCH	ANGLE MAIN-BR.1		120.0
		MAIN-BR.2		120.0
		FILLET RAD.	0.2000E-01	
		LOSS MAIN-BR.1	0.	
		MAIN-BR.2	0.	
		BR.1-MAIN	0.	
		BR.2-MAIN	0.	
		BR.2-BR.1	0.	

III- STROKE POSITION

TABLE IDENT.NR, 1

TYPE TIME DEPENDENT

TIME	STROKE
0.	1.000
0.2000	0.8000
0.4000	0.6000
0.6000	0.4000
0.8000	0.2000
1.000	0.

TABLE IDENT.NR, 2

TYPE TIME DEPENDENT

TIME	STROKE
0.5000	0.5000
0.6000	0.4000
0.8000	0.2000
1.000	0.

APPENDIX A

PROJECT 1177-2, TESTING EX. 1 PAGE NR. 7

IV-PERFORMANCE DATA

TABLE IDENT NR. 5

TYPE VALVE

RELATIVE VALUES	
STROKE	DISCH.COEFF.
0.	0.
0.2000	0.3300
0.4000	0.5500
0.6000	0.6900
0.8000	0.7700
1.000	0.8200

APPENDIX A

PROJECT 1177-2, TESTING EX. 1 PAGE NR. 1

CALCULATED PRESS.AND FLOW CONDITIONS

COND. NR.	FLOW CF/S	END 1 PRESS. LBS/FT^2	END 1 ENERGY HD. FT	END 2 PRESS. LBS/FT^2	END 2 ENERGY HD. FT
1	1388.0	-2975.3	529.99	16172.	336.83
2	694.02	17298.	296.63	9126.	296.08
3	694.02	-4779.8	73.228	1872.0	69.418
4	1388.0	16172.	336.83	16168.	336.77
5	694.02	17298.	296.63	9126.	296.08
6	694.02	-4779.8	73.228	1872.0	69.418

APPENDIX A

PROJECT 1177-2, TESTING EX. 2 PAGE NR. 1

CALCULATED PRESS.AND FLOW CONDITIONS

COND. NR.	FLOW CF/S	END 1 PRESS. LBS/FT^2	END 1 ENERGY HD. FT	END 2 PRESS. LBS/FT^2	END 2 ENERGY HD. FT
1	1303.8	-2404.6	529.99	18161.	359.57
2	570.85	19307.	322.54	13778.	322.16
3	570.85	-2224.9	65.711	1872.0	63.137
4	1303.8	18161.	359.57	18157.	359.51
5	732.92	18932.	325.06	9818.1	324.44
6	732.92	-5690.2	75.907	1872.0	71.656.

APPENDIX A

PROJECT 1177-2, TESTING EX. 3 PAGE NR. 1

CALCULATED PRESS.AND FLOW CONDITIONS

COND. NR.	FLOW CF/S	END 1 PRESS. LBS/FT²	END 1 ENERGY HD. FT	END 2 PRESS. LBS/FT²	END 2 ENERGY HD. FT
1	1303.8	-2404.6	529.99	18161.	359.57
2	-570.85	19307.	322.54	13778.	322.16
3	-570.85	-2224.9	65.711	1872.0	63.137
4	1303.8	18161.	359.57	18157.	359.51
5	732.92	18932.	325.06	9818.1	324.44
6	732.92	-5690.2	75.907	1872.0	71.656.