Section 5
Extended Transport Block

Following development of the original SWMM model, Water Resources Engineers (now Camp, Dresser and McKee) participated in a study of the proposed master plan for control of combined sewer overflows in San Francisco. In order to analyze the complex hydraulics of that system, they developed the WRE Transport Model (Shubinski and Roesner, 1973), one that solved the coupled complete St. Venant equations and accounts for phenomena such as backwater, looped connections, surcharging and pressure flow that were either not considered or treated in a very simplified manner in the original Transport Model (Section 6). Through subsequent work for EPA in other cities the WRE Transport Model was acquired for the SWMM package and became known as the Extended Transport Model or Extran. This model has few peers in its capacity for simulation of the hydraulics of urban drainage system and is probably the most sophisticated such model that is non-proprietary and available in the public domain. (Similar proprietary models do exist.) Extran capabilities are compared with those of Runoff and Transport in Table 4-3.

Extran has been part of the SWMM package since 1976. Extran originally contained both quantity and quality routines. However, the only (but very extensive) use for the model has been for hydraulic analysis, and the quality routing has been formally removed from the program. The state of the art in urban runoff quality modeling is such that adequate simulation of pollutographs may be performed using the simpler hydraulics of the Runoff and Transport Blocks.

The comprehensive SWMM Version 3 Extran documentation (Roesner et al., 1981) prepared by Camp, Dresser and McKee has been updated for new SWMM Version 4 features and is included as a separate addendum to this User's Manual (Roesner et al., 1987). Full details of the model are available therein. Interfacing between Extran and the remainder of SWMM is performed as described in Section 2.

<center>**Section 6**
Transport Block</center>

Block Description
Introduction

Routing through the sewer system may be accomplished in the Storm Water Management Model (SWMM) by the Transport Block. This block has the responsibility of coordinating not only routing of sewage quantities but also such functions as routing of quality parameters (subroutine QUAL), estimating dry-weather flow, DWF (subroutine FILTH), estimating infiltration (subroutine INFIL), and calling internal storage (subroutine TSTRDT). The relationships among the subroutines which make up the Transport Block are shown in Figure 6-1. The program is about 5,000 lines long, consisting of 24 subroutines and functions.

This section describes the Transport Block, provides instructions on data preparation, and furnishes examples of program usage. Instructions are provided for tasks requiring input data, namely: transport, internal storage, infiltration, and DWF.

Broad Description of Flow Routing

Differences in flow routing techniques among the Runoff, Transport and Extended Transport Blocks are described in Section 4 (e.g., Table 4-3); the techniques increase in complexity in the order just listed. A brief description of techniques used in the Transport Block follows.

To categorize a sewer system conveniently prior to flow routing, each component of the system is classified as a certain type of "element." All elements in combination form a conceptual representation of the system in a manner similar to that of links and nodes. Elements may be conduits, manholes, lift stations, overflow structures, or any other component of a real system. Conduits themselves may be of different element types depending upon their geometrical cross-section (e.g., circular, rectangular, horseshoe). A sequencing is first performed to order the numbered elements for computations. Flow routing then proceeds downstream through all elements during each increment in time until the storm hydrographs have been passed through the system.

The solution procedure is described in detail in the original SWMM documentation (Metcalf and Eddy et al., 1971a) and basically follows a kinematic wave approach in which disturbances are allowed to propagate only in the downstream direction. As a consequence, backwater effects are not modeled beyond the realm of a single conduit, and downstream conditions (e.g., tide gates, diversion structures) are assumed not to affect upstream computations. Systems that branch in the downstream direction can be modeled using "flow divider" elements to the extent that overflows, etc., are not affected by backwater conditions.
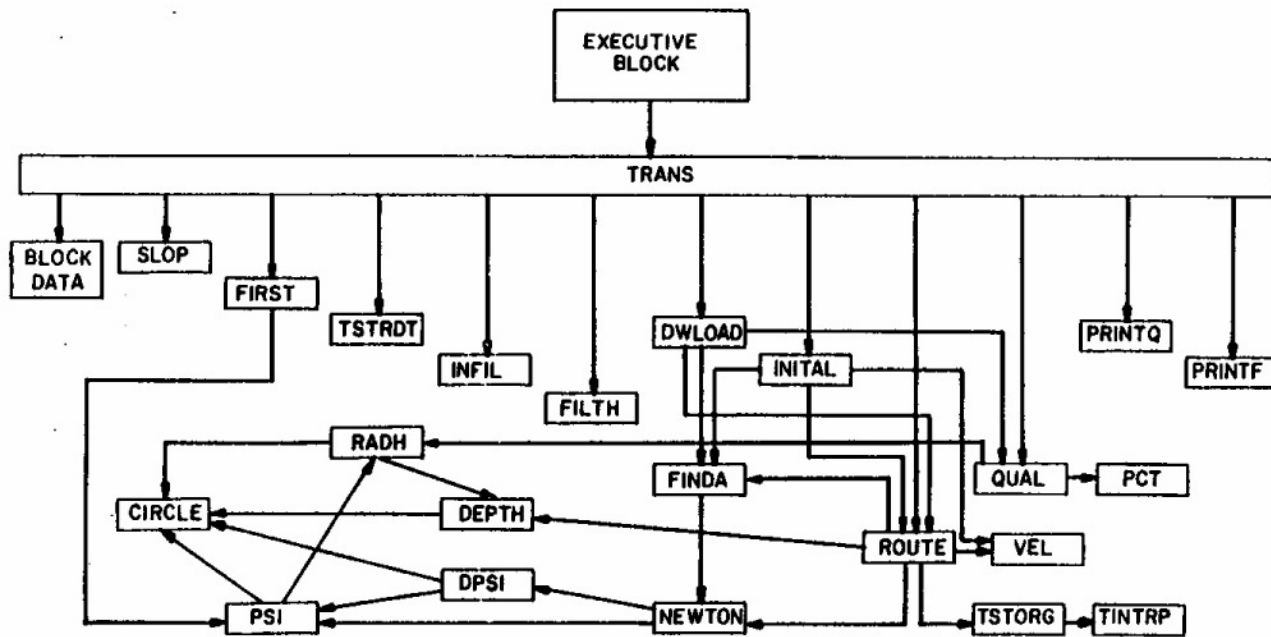
<center>208</center>

Figure 6-1. Structure of Transport Block subroutines.

Surcharging is modeled simply by storing excess flows (over and above the full flow conduit capacity) at the upstream manhole until capacity exists to accept the stored volume. Pressure-flow conditions are not explicitly modeled and no attempt is made to determine if ground surface flooding exists. A message is printed at each time step for each location at which surcharging occurs. The Transport Block has proven its ability to model accurately flows in most sewer systems, within the limitations discussed above, and as such it should be adequate for many applications. However, it will not accurately simulate systems with extensive interconnections or loops, systems that exhibit flow reversals, systems that exhibit significant backwater effects, or systems in which surcharging must be treated as a pressure-flow phenomenon; the Extended Transport Block should be used for these conditions.

An option in the program is the use of the internal storage model which acts as a transport element. It is a scaled down version of the Storage/Treatment Block and provides the possibility of storage routing of the storm (using the Puls method) for up to 30 locations within the sewer or drainage system. The program routes the flow through the storage unit for each time step based on the continuity equation in a manner analogous to flood routing through a reservoir. Extensive backwater conditions may thus be modeled by treating portions of the sewer system as a storage unit with a horizontal water surface.

Broad Description of Quality Routing

Up to four contaminants are handled by the Transport Block. Constituents may be introduced to the sewer system by any combination of four-means:

1) Storm-generated pollutographs computed by an upstream block[*] are transferred on an interface file (for description of the interface file see Section 2) to enter the system at designated inlet manholes.

2) Special storm-generated pollutographs may be entered using data group R1 of this block at designated inlet manholes.

3) Residual bottom sediment in the pipes may be resuspended due to the flushing action of the storm flows (Subroutines DWLOAD and QUAL).

4) For combined systems, constant dry-weather flow pollutographs may be entered at designated inlet manholes or generated by Subroutine FILTH.

Routing of the pollutants is then done for each time step by Subroutine QUAL. The maximum number of contaminants that can be routed is four. These may be selected arbitrarily from the input file, except that DWF can only be used to generate suspended solids, BOD_5 and total coliforms. The scour/deposition routines may be used for any constituent.

Program Operation

Most of the input is related to data needed to describe the particular sewer system being modeled (e.g., dimensions, slopes, roughnesses, etc.) and parameters needed to solve the governing flow routing equations.

Following input of these data, the sewer elements are sequenced for computations. Geometric and flow parameters are then initialized. The various program parameters and initialized variables describing the elements are then printed. Parameters relating to the amount of data to be stored and printed out are also read.

---

[*] The Transport Block can receive inputs from the Runoff, Storage/Treatment, and Extran Blocks, and from the Transport Block itself.

If required, infiltration values and quality parameters will be calculated. Alternatively, user-supplied values may simply be input at any manhole location. If desired, Subroutine DWLOAD then initializes constituent depositions, and Subroutine INITAL initializes flows and pollutant concentrations in each element to values corresponding to a condition of dry-weather flow and infiltration only.

The main iterations of the program consist of an outer loop on time steps and an inner loop on element numbers in order to calculate flows and concentrations in all elements at each time step. Inlet hydrographs and pollutograph ordinates are read from the interface file and/or Transport data groups to permit linear interpolation for values at each time step prior to entering the loop of element numbers.

When in the loop on element numbers (with index I), the current sewer element through which flows are to be routed, indicated by the variable M, is determined from the vector JR(I). This array is calculated in subroutine SLOP in a manner to insure that prior to flow routing in a given element, all flows upstream will have been calculated.

When calculating flows in each element, the upstream flows are summed and added to surface runoff, dry-weather flow (DWF), and infiltration entering at that element. These latter three quantities are allowed to enter the system only at non-conduits (e.g., manholes, flow dividers). If the element is a conduit, a check for surcharging is made. If the inflow exceeds the conduit capacity, excess flow is stored at the element just upstream (usually a manhole) and the conduit is assumed to operate at full-flow capacity until the excess flow can be transmitted. A final surcharging summary is printed.

A simple hydraulic design routine is available at this point. If desired (NDESN = 1), when a surcharge condition is encountered, the conduit will be increased in size in standard 6-in. increments (standard for U.S. pipes) for circular pipes or in 6-in. width increments for rectangular conduits until capacity exists to accept the flow. (Conduits that are neither circular nor rectangular will be converted to circular if they need to be resized.) A message is printed indicating the resizing, and a table of final conduit dimensions is printed at the end of the simulation. This design operation will effectively eliminate surcharging but will also minimize in-system storage within manholes, etc. The net effect is to increase hydrograph peaks at the downstream end of the system. An obvious conflict can thus exist between controls aimed at curing in-system hydraulic problems and controls intended for pollution abatement procedures at the outfall by in-system storage.

Flows are routed through each element in Subroutine ROUTE and quality parameters are routed in Subroutine QUAL. When routing flows in conduits, ROUTE may be entered more than once depending upon the value of ITER, the number of iterations. It is necessary to iterate upon the solution in certain cases because of the implicit nature of calculating the energy grade line in ROUTE.

Upon completion of flow and quality routing at all time steps for all elements. TRANS then performs the task of outputting the various data. Hydrograph and pollutograph ordinates for any specified outfall point(s) may be written onto an interface file for further use by the Executive Block, and Subroutine PRINTQ (or PRINTF for flows only) is then called for printing outflows for any of up to 80 desired elements.

Off-Line Files

The Transport Block uses two scratch data sets, NSCRAT(1) and NSCRAT(2), for storage of input and output hydrographs and pollutographs prior to printing. These are specified in the Executive Block. No input data will be sought from the interface file if JIN = 0, and no output will be placed on the file if JOUT = 0.

Organization of Data Input

Instructions for data preparation for the Transport Block have been divided along the lines of the major components for clarity of the presentation. These components are: (1) Transport, (2) Quality, (3) Internal Storage, (4) Infiltration and (5) Dry-Weather Flow. All data enter the Transport Block through one of these components. The typical data deck setup for the complete Transport Block is shown in Table 6-1. Transport data describe the physical characteristics of the conveyance system. Quality data identify pollutants to be routed and their characteristics. Internal storage data describe a particular type of Transport element. Infiltration and DWF data describe the necessary drainage area characteristics to permit the computation of the respective inflow quantities and qualities. Data line preparation and instructions for the complete Transport Block are given at the end of the Section 6 text in Table 6-7.

Table 6-1. Data for the Transport Block

Data Group	Data Type
A1	Title
B1-D3	Executive Data
C1	NKLASS, KPRINT
D1-D9	New Element Data
E1	Element Data
E2-E4	Cross Section Data
F1	Pollutant Data
G1-G5	Storage Element Data
H1-I1	Element I/O Data
J1-J2	Element Print Data
K1-K2	Infiltration Data
L1-Q1	Dry-Weather Flow Data
R1	User-Input Hydrographs and Pollutographs

Transport Data
Categories of Data
Use of the Transport Block involves three primary steps:

1) Preparation of theoretical data for use by subroutines engaged in hydraulic calculations in the block.
2) Preparation of physical data describing the sewer system.
3) Generation of inlet hydrographs and pollutographs required as input to the Transport Block and computational controls.

Data for Step 1 are supplied with the SWMM program for 16 different conduit shapes, and it will only be necessary for the user to generate supplemental data in special instances. These instances will occur only when conduit sections of very unusual geometry are incorporated into the sewer system. Generation of such data is discussed below.

The primary data requirements are for Step 2, the physical description of the sewer system, i.e., the tabulation of shape, dimension, slope, and roughness parameters, which is discussed in detail below.

The data for Step 3 may be generated manually, by an external block and by subroutines INFIL and FILTH.

Theoretical Data
TRANS reads the number and types of new conduit shapes found in the system. Only in the case of a very unusual shape should it become necessary to generate theoretical data to supplement the data supplied by the program. The required data describe flow-area relationships of conduits, as shown in Figure 6-2 through the parameters ANORM and QNORM. A similar depth-area relationship is also required using the parameter DNORM. Full details are provided by Metcalf and Eddy et al. (1971a).

The flow-area data are generated from Manning's equation, normalized by dividing by the corresponding equation for the conduit flowing full, denoted by the subscript f. Thus,

$$Q/Q_f = \frac{A\ R^{2/3}}{A_f \cdot R_f^{2/3}} = f(A/A_f) \quad (6\text{-}1)$$

where

Q = flow, cfs,
A = flow area, ft^2, and
R = hydraulic radius, ft.

For a given conduit shape (e.g., circular, rectangular, horseshoe), the hydraulic radius is a unique function of the area of flow; hence, Q/Q_f (interpolated between values of QNORM) is a function only of A/A_f (interpolated between values of ANORM). This function is tabulated for circular conduits in Appendix A of Chow (1959), for example, and on page 443 of Davis (1952) for a Boston horseshoe section. It is shown in graphical form for several conduit shapes in Chapter XI of Metcalf and Eddy (1914) from which some data supplied with this program have been generated. Swamee et al. (1987) provide analytical functions for four shapes. A listing of the conduit shapes supplied with the Transport Block as well as all other element types is given in
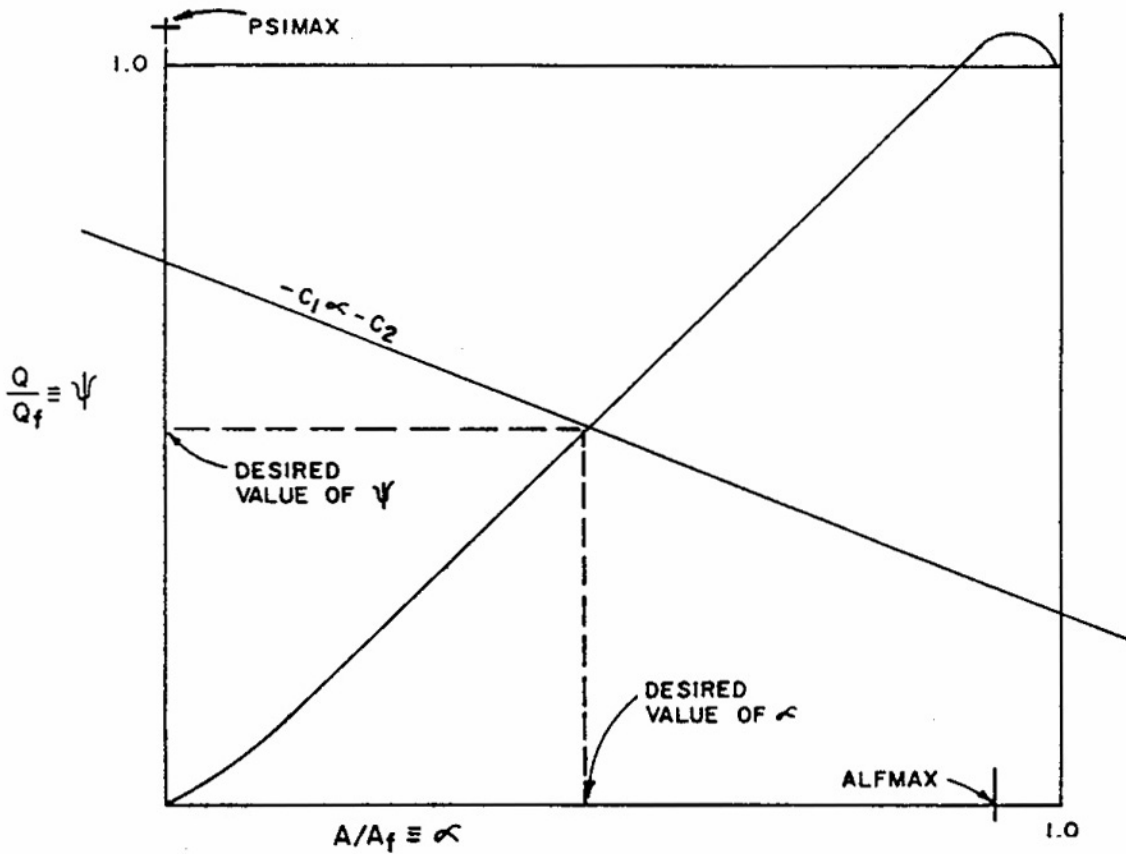
Figure 6-2. The intersection of the straight line and the normalized flow-area curve as determined in route. The ψ-$\blacktriangle$ curve is formed by straight line segments delineated by the variables ANORM and QNORM, for conduits with a tabular Q-A relationship. Q denotes flow, A denotes area, and the subscript f denotes values at full-flow. The line $-C_1$ $\blacktriangle$ $-C_2$ is formed by the program from the continuity equation.
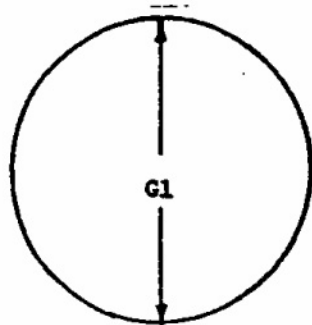
Table 6-2. The conduits are illustrated in Figure 6-3. If y = depth of flow, values of y/y_f corresponding to A/A_f (ANORM) are tabulated as the variable DNORM.

It will often be satisfactory to represent a shape not included in Table 6-2 by one of similar geometry. This use of "equivalent" sewer sections will avoid the problem of generating flow-area and depth-area data. An equivalent section is defined as a conduit shape from Table 6-2 whose dimensions are such that its cross-sectional area and the area of the actual conduit are equal. Only very small errors should result from the flow routing when this is done.
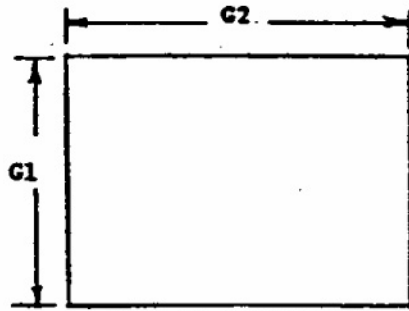
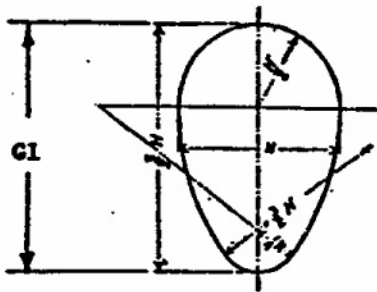Table 6-2. Different Element Types Supplied with the Transport Block

NTYPE	Conduits/Channels
1	Circular
2	Rectangular
3	Phillips standard egg shape
4	Boston horseshoe
5	Gothic
6	Catenary
7	Louisville semielliptic
8	Basket-handle
9	Semi-circular
10	Modified basket-handle
11	Rectangular, triangular bottom
12	Rectangular, round bottom
13	Trapezoid
14	Parabolic
15	Power Function
16	HEC-2 Format - Natural Channel
17, 18	User supplied
	Non-Conduits
19	Manhole
20	Lift station
21	Flow divider
22	Storage unit
23	Flow divider - weir
24	Flow divider
25	Backwater element

Figure 6-3. Sewer cross-sections.
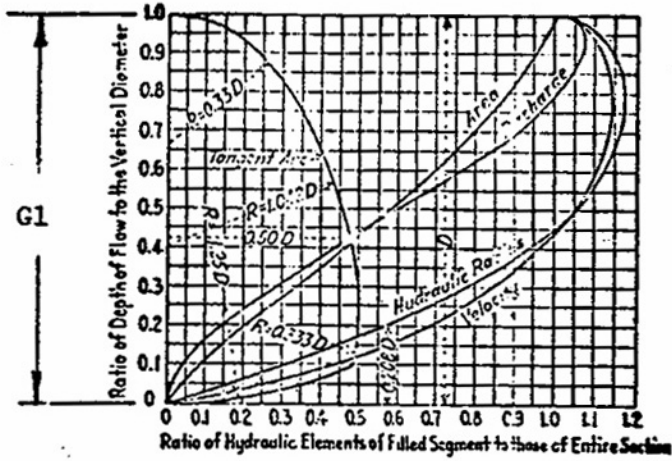
Type 7: Louisville Semielliptic
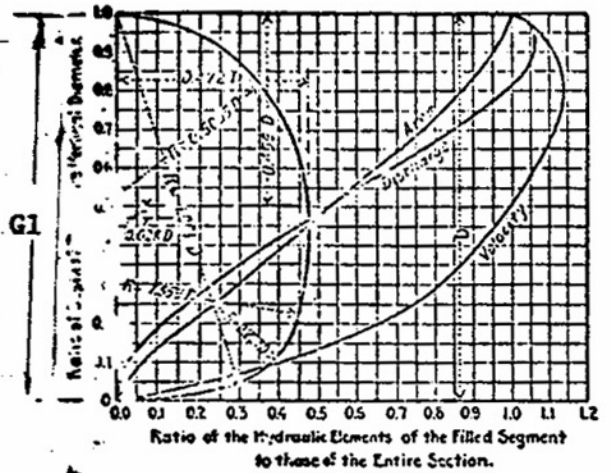
Type 8: Basket-handle
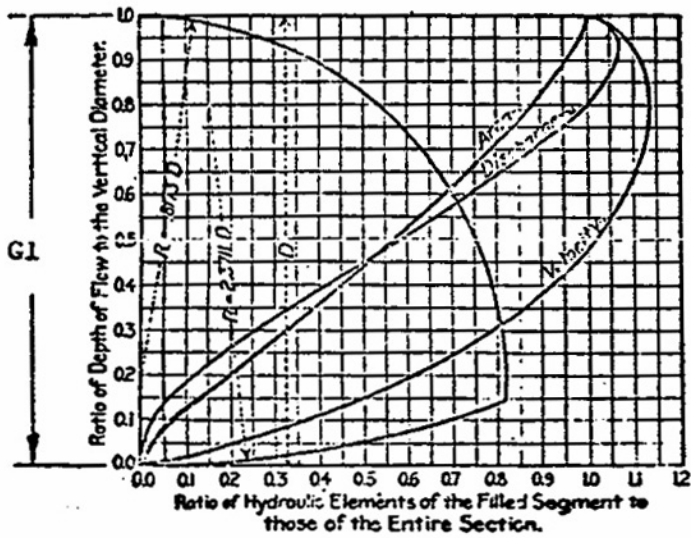
Type 9: Semi-circular
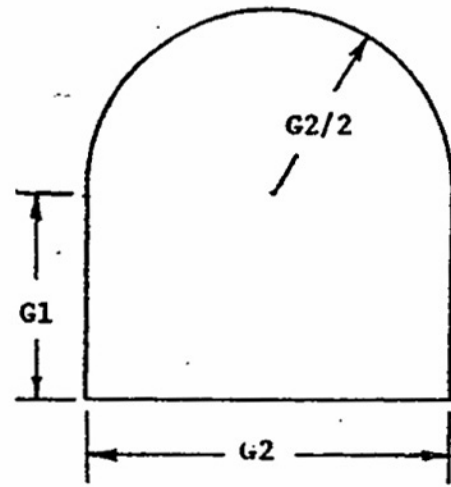
Type 10: Modified Basket-handle
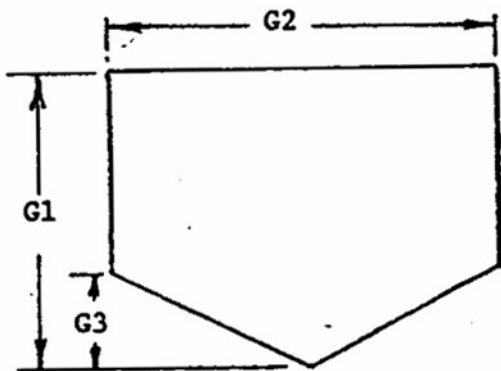
Type 11: Rectangular, Triangular Bottom
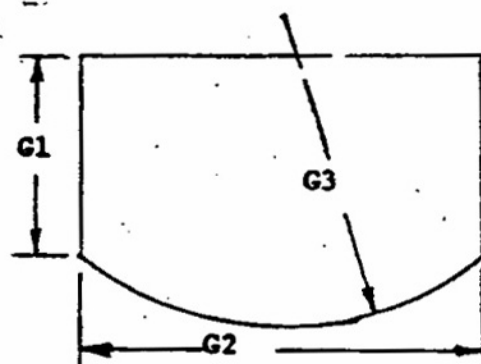
Type 12: Rectangular, Round Bottom

Figure 6-3. Continued.

217

TYPE 13: TRAPEZOID

Type 14: Parabolic, and
Type 15: Power Function
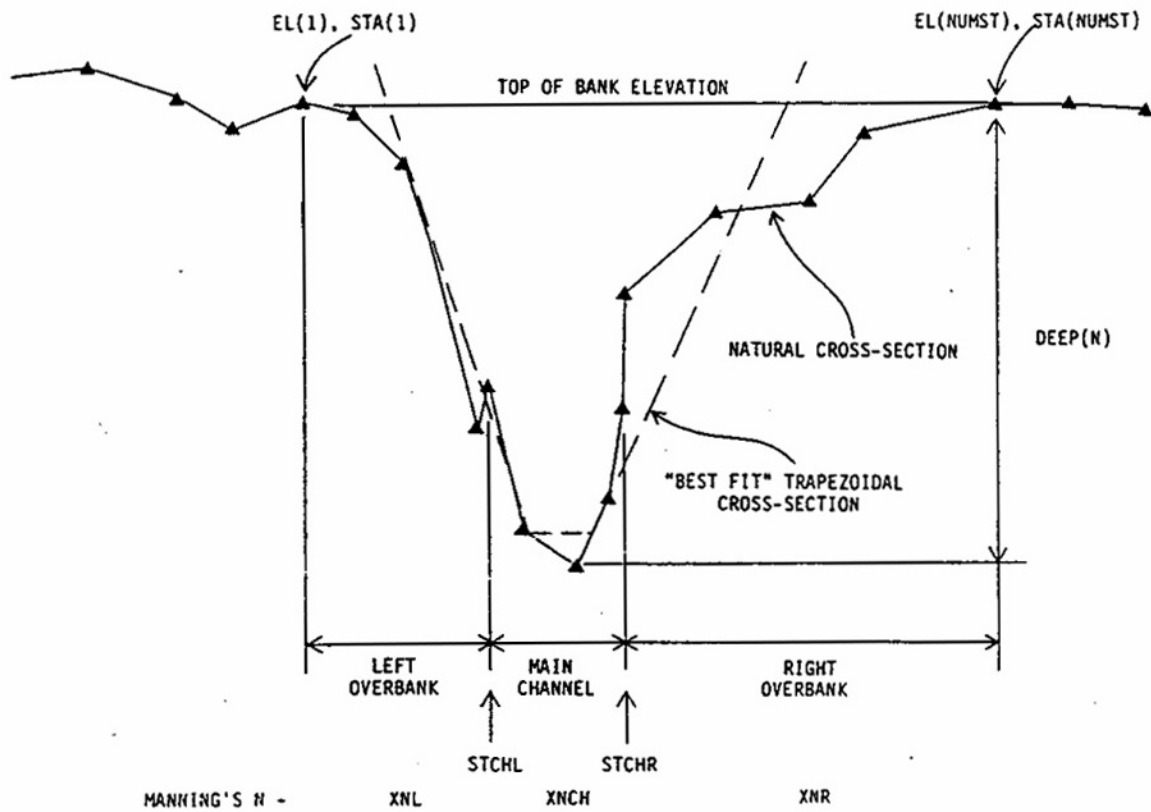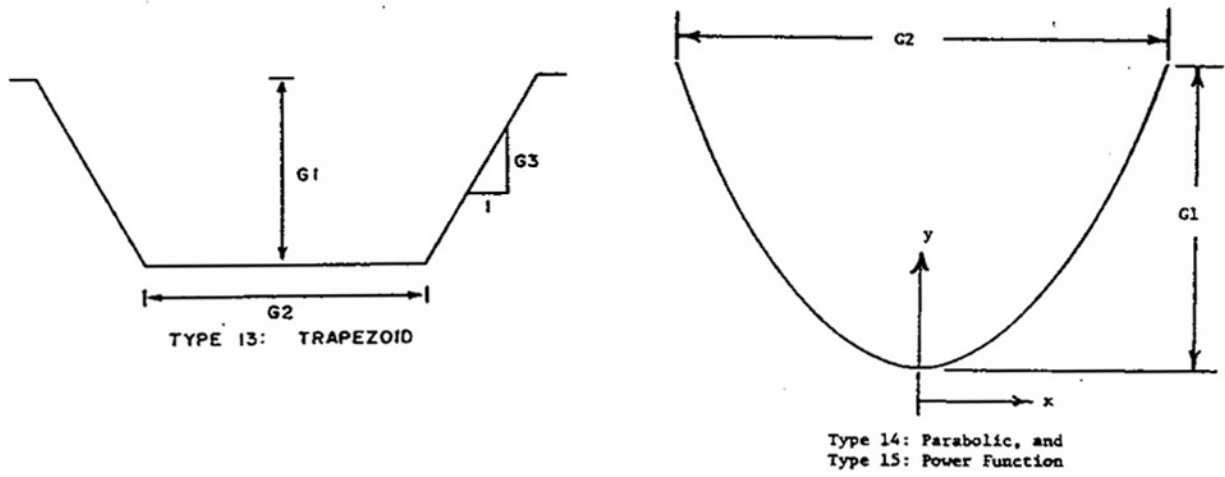
Type 16

Definition Sketch of an Irregular Cross-Section.

Figure 6-3. Continued.

218

If it is desired to have the exact flow-area and depth-area relationships, then the product $AR^{2/3}$ must be found as a function of area. In general, the mathematical description of the shape will be complex and the task is most easily carried out graphically. Areas may be planimetered, and the wetted perimeter measured to determine R. In addition, the depth may be measured with a scale. The required flow-area relationship of equation 6-1 may then be tabulated as can the depth-area relationship. The number of points on the flow-area and depth-area curves required to describe the curves is an input variable (MM and NN, respectively). Note that the normalized flows (QNORM) and depths (DNORM) must be tabulated at points corresponding to MM-1 and NN-1, respectively, equal divisions of the normalized area axis (ANORM). If desired, the routing parameters stored in the program may be listed by specifying KPRINT = 1 on data line A1. The four pages of output are seldom necessary during the simulations, however.

Physical Representation of the Sewer System
General
These data are the different element types of the sewer system and their physical descriptions. The system must first be descritized as a network of conduit lengths, joined at manholes (or other non-conduits). In addition, either real or hypothetical manholes should delineate significant changes in conduit geometry, dimensions, slope, or roughness. Finally, inflows to the system (i.e., stormwater, wastewater, and infiltration) are allowed to enter only at manholes (or other non-conduits). Thus, manholes must be located at points corresponding to inlet points for hydrographs generated by an external block and input points specified in Subroutines FILTH and INFIL. In general, the task of identifying elements of the sewer system will be done most conveniently in conjunction with the preparation of data for these other subroutines, especially the Runoff Block.

Each element (conduit or non-conduit) must be identified with a number which may range from 1 to 10,000. They need not be sequential or continuous. Experience has shown that a schematic map showing the complete sewer network and the numbering system will be very useful for debugging and identification purposes. It is difficult to rely upon detailed (and often cluttered) sewer plans alone.

Description of Conduits
The 16 conduit shapes supplied with SWMM are shown in Figure 6-3. For each shape, the required dimensions are illustrated in the figure and specified in Table 6-3. In addition, Table 6-3 gives the formula for calculating the total cross-sectional area of the conduit.

Usually, the shape and dimensions of the conduit will be indicated on plans. It is then a simple matter to refer to Figure 6-3 for the proper conduit type and dimensions. If the shape does not correspond to any supplied by the program, it will ordinarily suffice to choose a shape corresponding most nearly to the one in question. For example, an inverted egg can be reasonably approximated by a catenary section. The dimensions of the substitute shape should be chosen so that the area of the substitute conduit and that of the actual conduit are the same. This is facilitated by Table 6-3, in which the area is given as a function of conduit dimensions. If desired, the flow-depth area parameters for up to two additional conduit shapes may be read in at the beginning of the program as discussed previously. (See also Data Groups D1-D9.)

Table 6-3. Summary of Area Relationships and Required Conduit Dimensions.[a]

NTYPE	Shape	Area	Required Dimensions
1	Circular	$(\text{pi}/4)\times G1^2$	GEOM1 = Diameter
2	Rectangular	$G1\times G2$	GEOM1 = Height GEOM2 = Width
3	Egg-shaped	$0.5105\times G1^2$	GEOM1 = Height
4	Horseshoe	$0.829\times G1^2$	GEOM1 = Height
5	Gothic	$0.655\times G1^2$	GEOM1 = Height
6	Catenary	$0.703\times G1^2$	GEOM1 = Height
7	Semi-elliptic	$0.785\times G1^2$	GEOM1 = Height
8	Basket-handle	$0.786\times G1^2$	GEOM1 = Height
9	Semi-circular	$1.270\times G1^2$	GEOM1 = Height
10	Modified basket-handle	$G2[G1+(\text{pi}/8)G2]$	GEOM1 = Side height GEOM2 = Width
11	Rectangular triangular bottom	$G2(G1-Ge/2)$	GEOM1 = Height GEOM2 = Width GEOM3 = Invert height
12	Rectangular, round bottom	$G1\times G2 + G3^2/2\times(\phi-\sin\phi)$ $\phi = 2\times\sin^{-1}[G2/(2\times G3)]$	GEOM1 = Height GEOM2 = Width GEOM3 = Invert radius
13	Trapezoidal channel	$G1(G2+G1/G3)$	GEOM1 = Depth GEOM2 = Bottom width GEOM3 = Side slope (vertical/horizontal)
14	Parabolic channel	$(2/3)\times G1\times G2$	GEOM1 = Depth GEOM2 = Top width
15	Power function channel: $x^{G3} = k\times y$	$[G3/(1+G3)]\times G1\times G2$ $k = (1/G1)\times(G2/2)^{G3}$	GEOM1 = Depth GEOM2 = Top width GEOM3 = Exponent of power function

[a]Refer for Figure 6-3 for definition of dimensions G1, G2, and G3.
Note that G1 = GEOM1, G2 = GEOM2, G3 = GEOM3.

Occasionally, the conduit dimensions and area may be given, but the shape not specified. It will sometimes be possible to deduce the shape from the given information. For example, a conduit may have an area of 4.58 ft^2 (0.425 m^2) and dimensions of 2 ft by 3 ft (0.6 by 1.0 m). First, assume that the 2-ft dimension is the width and the 3-ft dimension is the depth of the conduit. Second, note from Figure 6-3 that the ratio of depth to width for an egg-shaped conduit is 1.5:1. Finally, the area of an egg-shaped conduit of 3 foot depth is $0.5105 \times 9 = 4.59$ ft^2 (0.426 m^2). It is concluded that the conduit should be type 3 with GEOM1 = 3 ft.

Because of the limits on the size of the computer program, it will usually not be possible to model every conduit in the drainage basin. Consequently, aggregation of individual conduits into longer ones will usually be the rule. Average slopes and sizes may be used provided that the flow capacity of the aggregate conduit is not significantly less than that of any portion of the real system. This is to avoid simulated surcharge conditions that would not occur in reality. In general, flow calculations are relatively insensitive to conduit lengths, although with conduits over 4000 to 5000 ft (1200 m and 1500 m) long some loss of routing accuracy will result. This is caused primarily when a large inflow enters a dry or nearly dry pipe, often at the beginning of the simulation. A non-convergence error message will be printed, but the resultant error is seldom significant. Conduit lengths should always be separated by manholes (or other non-conduit type elements). The conduit length should be measured from the center of the adjacent manholes. A further means of simulating large systems lies in simulating different portions with separate Transport runs and combining the results using the Combine Block (see Section 3).

Values of Manning's roughness may be known by engineers familiar with the sewer system. Otherwise, they may be estimated from tables in many engineering references (e.g., Chow, 1959; ASCE-WPCF, 1969) as a function of the construction material and sewer conditions. The value may be adjusted to account for losses not considered in the routing procedure (e.g., head losses in manholes or other structures, roots, obstructions). However, the flow routing is relatively insensitive to small changes in Manning's n.

Description of Non-Conduits

Types. The sewer system consists of many different structures, each with its own hydraulic properties. Elements types 19 through 25 (denoted by parameter NTYPE on group E1) are designed to simulate such structures. Data requirements for these elements are given in Table 6-4. Brief descriptions of these elements follow.

Manholes (NTYPE = 19). No physical data are required for manholes except their numbers and upstream element numbers. Note that the number of upstream elements is limited to three. If more than three branches of the system should join at a point, two manholes could be placed in series, allowing a total of five branches to joint at that point, etc. Flow routing is accomplished in manholes by specifying that the outflow equals the sum of the inflows.

As an alternative to the use of the more detailed infiltration (INFIL) and dry-weather flow (FILTH) routines described later, flow and quality constituents may be input at manholes to simulate baseflow conditions. This input is constant over time and is allowed only at manholes and at no other element types.

Table 6-4. Parameters Required for Non-Conduits

NTYPE	Description	DIST	GEOM1	SLOPE	ROUGH	GEOM2	BARREL	GEOM3
19	Manhome	Constant inflow, cfs, [m³/sec].	Const. inflow concen. of pollutant 1.[a]	Const. inflow concen. of pollutant 2.[a]	Const. inflow concen. of pollutant 3.[a]	Const. inflow concen. of pollutant 4.[a]	N.R.[b]	N.R.
20	Lift station	Pumping rate, assumed constant, cfs, [³/sec].	Volume in wet well at which pumps will start, ft³, [m³].	N.R.	N.R.	N.R.	N.R.	N.R.
21	Flow divider	N.R.	Maximum undiverted flow. Inflow in excess of this value is diverted, cfs, [m³/sec].	N.R.	N.R.	N.R.	N.R.	Number of element into which flows the undiverted flow (include decimal point).
22	Storage unit[c]	N.R.	N.R.	N.R.	N.R.	N.R.	N.R.	N.R.
23	Flow divider	Maximum inflow without flow over the weir, cfs, [m³/sec].	Weir height, above zero flow depth, ft, [m].	Maximum inflow through whole structure, cfs, [m³/sec].	Weir constant times weir length, ft, [m].	Depth in structure at time of maximum inflow, ft, [m].	N.R.	Number of element into which flows the undiverted flow (weir flow is the diverted flow).
24	Flow divider	N.R.	N.R. (assigned in program)	N.R.	N.R.	N.R.	N.R.	Number of element into which flows the undiverted flow.
25	Backwater element	N.R.	N.R.	N.R.	N.R.	N.R.	N.R.	Element number of downstream storage flow.

[a]Units according to NDIM, Card group F1.

[b]N.R. – Not Required.

[c]Additional parameters are read in subsequently in Card groups G1-G5.

NOTE: All elements require an element under (NOE), three upstream element numbers (NUE), and type (NTYPE). Parameters for conduits are defined in Table 6-2.

Lift Stations (NTYPE = 20). The data requirements for lift stations are given in Table 6-4. It is assumed that the force main will remain full when the pump is not operating, resulting in no time

222

delay in the flow routing (i.e., no time is required to fill the force main when the pump starts, and the force main volume is not accounted for during pollutant routing). When the volume of sewage in the wet well reaches its specified capacity, the pumps begin to operate at a constant rate. This continues until the wet well volume equals zero. (Two-stage pumping may be simulated using a storage element.)

Flow Dividers (NTYPE = 21 and 24). The routing procedure through these elements is explained in the discussion below. Typical uses are given below.

1) Simple diversion structure – A type 21 flow divider may be used to model a diversion structure in which none of the flow is diverted until it reaches a specified value (GEOM1). When the inflow is above this value, the non-diverted flow (QO1) remains constant at its capacity, GEOM1, and the surplus flow (QO2) is diverted.

2) Cunnette section – A type 24 flow divider may be used to model a downstream cunnette section. The cunnette section is considered as a separate circular conduit to be placed parallel to the primary conduit as shown in Figure 6-4. In order to model the cunnette as a semi-circle, the separate circular conduit is given a diameter (GEOM1) so that its area will be twice that of the actual total cunnette flow area. (The distance, slope and roughness will be the same as for the primary conduit.) A type 24 flow divider is then the upstream element common to both conduits, as shown in Figure 6-4. (The program assigns a value of GEOM1 of the flow divider equal to half the full flow capacity of the circular pipe simulating the cunnette so that it has the hydraulic characteristics of a semi-circle.) Any flow higher than GEOM1 will be diverted to the primary conduit. Note that the parameter GEOM3 of the flow divider will be the element number assigned to the cunnette section. Note further that the element downstream from the two parallel conduits must list them both as upstream elements.

3) Overbank flow – A type 21 flow divider can be used to simulate flow into a main channel (undiverted flow) and into a parallel overbank channel for simulation of flooded conditions. Parameter GEOM1 would be set equal to the main channel capacity. The channel could be of any shape although two trapezoidal channels might be most appropriate for many natural configurations.

Routing at Flow Dividers (NTYPE = 21 and 24). Both types will divide the inflow, QI, into two outflows, QO1 and QO2. The divider then acts as follows:

$$\text{For } 0 \le QI \le GEOM1, \qquad \begin{aligned} QO1 &= QI \\ QO2 &= 0.0 \end{aligned}$$

$$\text{For } GEOM1 \le QI, \qquad \begin{aligned} QO1 &= GEOM1 \\ QO2 &= QI - GEOM1 \end{aligned}$$

(6-2)

The undiverted outflow, QO1, will flow into the downstream element denoted by GEOM3. (The element into which QO2 flows does not need to be specified.)

223

SECTION OF SEWER
WITH CUNNETTE

PRIMARY CONDUIT   PRIMARY CONDUIT

CUNNETTE (TYPE I)   CUNNETTE (TYPE I)

FLOW DIVIDER (TYPE 24)
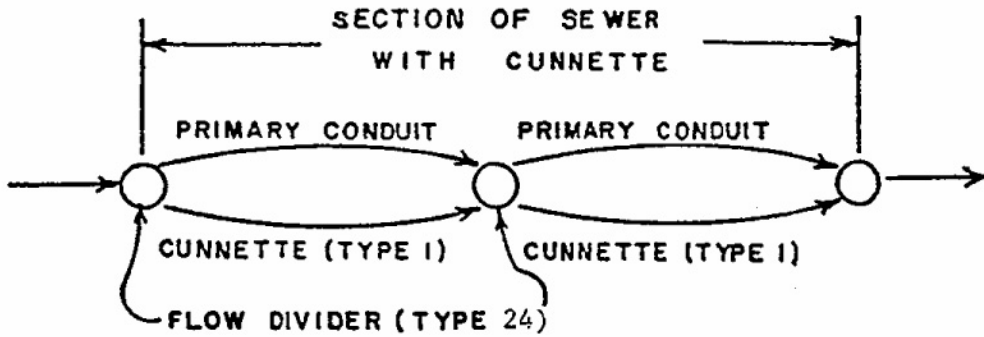
a.   SCHEMATIC OF HYPOTHETICAL FLOW DIVISION

CONDUIT WITH
CUNNETTE

=

PRIMARY
CONDUIT

+
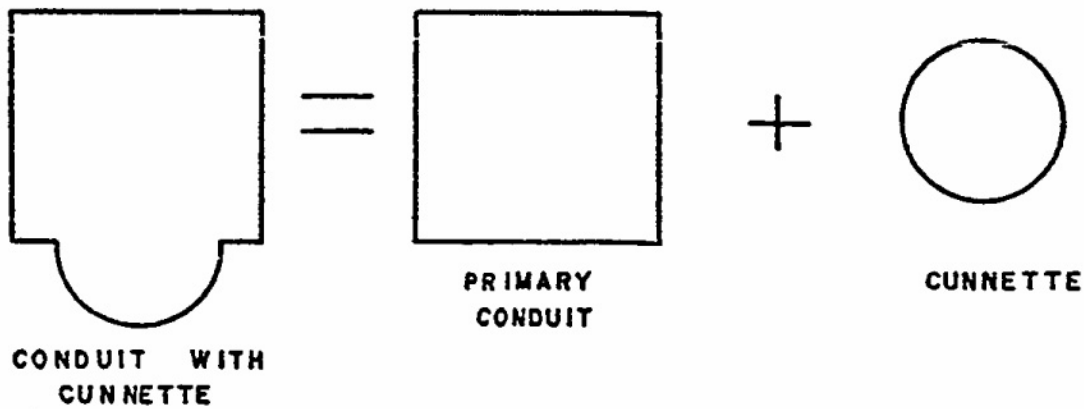
CUNNETTE

Figure 6-4. Cunnette section.

Flow Divider (NTYPE = 23). This element is used to model a weir-type diversion structure in which a linear relationship can adequately relate the flow rate and the depth of flow into the weir structure. Input parameters are defined in Table 6-4. The weir constant, incorporated into the variable ROUGH, can be varied to account for the type of weir. Typical values of the weir constant are 3.3 for a broad crested weir and 4.1 for a side weir.

The flow divider behaves as a function of the inflow, QI, as follows:

For Q ≤ QI ≤ DIST,   QO1 = QI
                       QO2 = 0.0

(6-3)

For DIST ≤ QI,   QO1 and QO2 are computed as follows:

1) Compute depth of flow above the weir, DH, assuming a linear flow-depth relationship:

$$DH = (QI\text{-}DIST)\cdot(GEOM2\text{-}GEOM1)/(SLOPE\text{-}DIST)$$

2) Compute the diverted flow from the weir formula:

$$Q02 = ROUGH \diamondsuit DH^{1.5}$$

3) Compute the undiverted flow:

$$QO1 = QI \text{-} QO2$$

Storage Unit (NTYPE 22). This element is specified only when internal storage computations are required. Internal storage is modeled in a manner similar to a detention unit of the Storage/ Treatment Block, using Puls routing for hydrographs and simulating quality processes as a completely mixed reactor. However, unlike a detention unit in the S/T Block, pollutant removal may be simulated only by decay. Data requirements are described on data groups G1 to G5.

Up to 30 storage units may be placed anywhere in the sewer system where appreciable storage may exist, such as at an outflow or diversion structure, or to simulate detention storage in a stormwater management system. It should be noted that the storage area or "reservoir" may consist of a portion of the sewer system itself, and area-depth-discharge relationships developed accordingly.

Backwater Element (NTYPE = 25). This element may be used to approximate backwater conditions in a series of conduits due to a flow control structure downstream. The situation is modeled in a manner analogous to reservoir flood routing as follows:
1) A storage element (NTYPE 22) is placed at the location of the control structure. The type of storage element will depend upon the structure (e.g., weir, orifice). One inflow to this storage element is then from the conduit just upstream.
2) If the water surface is extended horizontally upstream from the flow control structure at the time of maximum depth at the structure, it will intersect the invert slope of the sewer at a point corresponding to the assumed maximum length of backwater. The reach between this point and the structure may encompass several conduit lengths. A backwater element (NTYPE 25) is placed at this point of maximum backwater, in place of a manhole, for instance.

3) The backwater element then diverts flow directly into the storage element depending upon the volume of water (and hence, the length of backwater) in the storage element. If the backwater extends all the way to the backwater element, the total flow is diverted to the storage element; none is diverted to the conduits.

4) The amount of diverted flow (QO1) is assumed to be directly proportional to the length of the backwater. The storage area in reality consists of the conduits. Since most conduits can be assumed to have a constant width, on the average, the backwater length is assumed to be proportional to the square root of the current storage volume, obtained from the storage routine.

5) The parameter GEOM3 of the backwater element must contain the element number of the downstream storage unit.

6) Parameters for the storage element are read in as usual. The depth-area values will correspond to the storage area of the upstream conduits. The storage unit must list the backwater element as one of its upstream elements, as well as the conduit immediately upstream.

7) At each time step, the backwater element computes the ratio of current to maximum storage volume in the downstream storage element. Call this ratio r. Then

$$QO1 = QI \diamond r^{1/2}$$

and   (6-4)

$$QO2 = QI - QO1$$

where

$$
\begin{aligned}
QO1 &= \text{flow directly into storage unit,} \\
QO2 &= \text{flow into intermediate conduits, and} \\
QI &= \text{inflow to backwater element.}
\end{aligned}
$$

Input Data and Computational Controls

Options

The basic input data, hydrographs and pollutographs, are generated outside of the Transport Block. However, certain operational controls are available within Transport.

Time Step (DT)

The size of the time step may be chosen arbitrarily and does not have to correspond to that of the preceding block. If it does not, input hydrograph and pollutograph ordinates will be estimated by linear interpolation of the input time series on the interface file. However, awkward interpolation effects can be avoided if the Transport time step is an integer multiple or integer fraction (e.g., 1/2, 1/3) of the time step of the upstream block. If the upstream block uses a variable time step (e.g., the Runoff Block), the shortest of the variable time steps should be used to compute the integer fraction. Transport uses a constant time step.

In tests of sensitivity (Metcalf and Eddy et al., 1971a), it was found that except for very small values of DT (10 seconds), the output from Transport is insensitive to the length of the time step. Between values of two minutes and 30 minutes, hydrograph ordinates varied by less than one percent. For extremely short time step values, the peak flow moved downstream faster and never attained the maximum value that it had with a DT of two minutes and longer. Within the range commonly needed (two minutes to 30 minutes), the choice of time step will not significantly affect results. However, continuity errors can occasionally arise if the time step is longer than about two times the travel time through any conduit.

Number of Time Steps (NDT)

The number of time steps is not restricted. The program will use the number input in Transport (NDT) or the number used by the preceding module, whichever gives the shorter simulation time.

Choice of Number of Iterations (NITER)

The purpose of iterations in the computations is to eliminate flow oscillations in the output. Flatter pipe slopes (less than 0.001 ft/ft) require iterations of the flow routing portion of the Transport Model to help dampen these oscillations. Four iterations have proven to be sufficient in most cases.

Allowable Convergence Error (EPSIL)

Convergence of the flow routing procedure should not be any problem, and the default value of EPSIL, 0.0001, may be used. It will provide sufficient accuracy and result in only a very minimal increase in computer time over larger values. The only convergence problems that may exist can occur when flow enters a dry conduit. For instance, this could occur at the beginning of a storm in a sewer with little or no baseflow. Messages to this effect will be printed if parameter NPRINT = 0. These may almost always be ignored since the default options in Subroutine ROUTE will continue program execution and only result in a very small error in continuity (a fraction of a percent).

Alternate Hydrograph and Pollutograph Inputs

Hydrograph and pollutographs may be entered from an interface file (e.g., as generated in the Runoff Block) and/or entered manually using data groups I1 and R1. Parameters NCNTRL and NINPUT are set accordingly. Note that input from both a Transport input file and interface file may be performed simultaneously. If, for some reason, input from data group R1 will not suffice, an interface file containing the specified input values could be created according to the specifications found in Section 2 and specified as an input file to Transport in place of, say, a file generated by the Runoff Block. The format of such a file is described in the Tables 2-2 and 2-3. In this manner, a user may use an alternative program for generation of the inlet hydrograph, followed by Transport for routing.

Quality
Constituents

Up to four pollutants may be arbitrarily chosen for input and routing by the Transport Block. Although these would often be chosen from the group (up to ten) supplied by the Runoff Block (or another preceding block), they do not have to be since data group input may be used in addition to the interface file. If the same pollutant is entered from both the interface file and from the data group R1, the description (name, name of units, type of units) from the interface file must be used. If the pollutant is entered only from the data group, this description must be supplied on data group F1. Further information on pollutant description, including naming and units, is contained in Section 4.

Decay

Each pollutant may be subjected to a first order decay during the routing process by supplying a first order decay coefficient, DECAY (based on natural logarithms or base e). Although travel time through most sewer systems is short enough so that decay is seldom important, the user could supply, for example, a deoxygenation coefficient, K_1, for BOD if desired. Non-conservative pollutants are not linked. The decay of one has no effect on any other.

Routing

Routing of quality parameters is performed by using the integral solution for the output from a completely mixed conduit volume (Medina et al., 1981). See Appendix IX for a derivation. Although this tends to introduce artificial dispersion of concentration profiles, it is the most convenient way in which to introduce new loadings at manholes along the system, as well as to facilitate scour and deposition calculations. The quality routing procedure is not subject to calibration directly. However, the routing becomes closer to pure advection (plug flow) as the number of elements is increased.

Scour and Deposition

The basis for these procedures is described in Appendix VI. Each pollutant is assigned a specific gravity (SPG) and particle size distribution, assumed to apply throughout the drainage system regardless of the source of the pollutant, e.g., stormwater or dry-weather flow. If the specific gravity is less than or equal to 1.0, the pollutant is considered to be entirely suspended (or dissolved) and not subject to scour and deposition. If all calibration is to be performed using Runoff Block buildup-washoff parameters, for instance, it may be desirable to avoid the complexity of simulating scour, a second real but largely unknown source.

Typical particle size distributions (and interpretation of input parameters PSIZE and PGR) are illustrated in Figure 6-5. Such information should be collected first hand at each catchment; secondary sources such as Sartor and Boyd (1972), Shaheen (1975), Manning et al. (1977) and Pisano et al. (1979) should be used only if local data are not available. During the simulation scour and deposition are simulated using Shield's criterion to determine the critical diameter for incipient motion and deposition (see Appendix VI). The kinematic viscosity of water (GNU on data line B2) is a function of temperature and used to calculate the boundary Reynolds number on Shield's diagram (Graf, 1971; Vanoni, 1975). For each conduit, the critical diameter is determined as a function of velocity, roughness and specific gravity. At the same

time, the <u>maximum</u> diameter of the <u>suspended</u> fraction and the <u>minimum</u> diameter of the <u>settled</u> fraction is
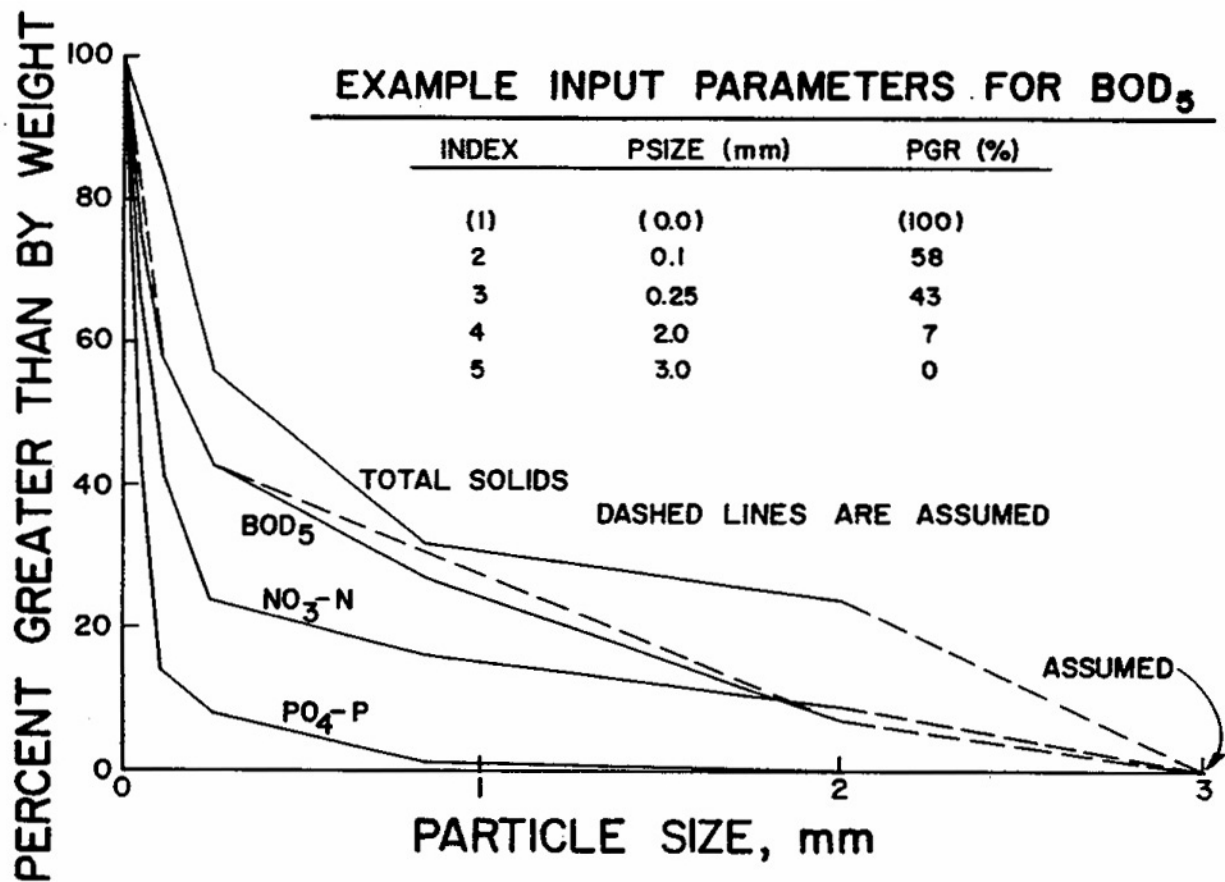
Figure 6-5. Example particle size distributions for pollutants found on street surfaces (after Sartor and Boyd, 1972, p. 146).

maintained. If the critical diameter is less than the maximum of the suspended material, more is settled; the settled mass is determined by multiplying by a fraction determined from the particle size distribution (Appendix VI). Similarly, if the critical diameter is greater than the minimum of the settled material, more is suspended. The settled material is thus assumed to have the particle size distribution of the right hand tail of the total distribution (Figure 6-5), and the suspended material has the distribution of the left hand side.

Decreasing the specific gravity (downwards toward 1.0) increases the amount suspended and vice versa. As SPG closely approaches 1.0 the procedure becomes very sensitive to SPG since there is a division by SPG-1.0. Typical values of specific gravities of particulate matter in sewers range from 1.1 for volatile material to 2.7 for sand and grit. The realistic situation in which each particle size range may have its own specific gravity can be handled by the

Storage/Treatment Block, but not the Transport Block (except that up to four different pollutants may be simulated). Since it is only one parameter, calibration of the scour-deposition routine may be most easily calibrated using SPG. Alternatively, a greater percentage of large diameter material may be assigned a pollutant using the particle size distribution if, for instance, more deposition were desired.

Continuity of pollutant mass is maintained during scour and deposition. In addition, larger particles can settle upstream in flat conduits and be unavailable for downstream settling. However, no layering within the sediment bed is possible; a uniform distribution with sediment depth is assumed. An initial settled mass in each conduit is computed prior to the start of the simulation by running the routine for DWDAYS days (data line B2) prior to the storm event (or longer) simulation. This initial deposition is assumed to start with a clean bed.

Although this scour-deposition routine is a far cry from the detailed sewer sediment transport program developed by Sonnen (1977), it is reasonably simple, consistent and may be calibrated. And should the user desire, it may be bypassed (using SPG < 1.0), and all quality calibration performed in the Runoff Block.

Internal Storage
Steps
Use of the internal storage routine involves four basic steps. A somewhat more detailed data description may be found in the Storage/Treatment Block description in Section 7, and modeling techniques are described in Appendix IV.

Step 1. Call
The internal storage routine is called by subroutine TRANS when element NTYPE 19 is specified. No more than 30 storage locations may be specified in a single run.

Step 2. Storage Description: Part 1
Describe the manner in which the outlet depth-discharge relationship is given (set of data pairs, power equation or pumps). See Appendix IV and Section 7 for a more detailed description of this technique.

Step 3. Storage Description: Part 2
Describe the geometry of the unit with a set of depth-surface area volume data triplets and the depth-discharge relationship with data pairs or a power equation. See Appendix IV and Section 7 for more details.

Step 4. Initial Conditions
Describe the initial conditions of the unit with respect to volume and pollutant concentrations.

Infiltration Model
Description

The section on infiltration has been developed to help to estimate infiltration into a given sewer system based upon existing information about the sewer, its surrounding soil and groundwater, and precipitation. It should be borne in mind throughout that the accuracy of infiltration prediction is dependent upon the accuracy and extent of data descriptive of infiltration in the system being modeled.

Using these data, Subroutine INFIL is structured to provide estimates of average daily infiltration inflows at discrete locations along the trunk sewers of a given sewer system. A typical urban drainage basin in which infiltration might be estimated is shown in Figure 6-6.

Since the Transport Block's principal use has been mainly to simulate individual storms which cover a time period of less than a day, average daily estimates from INFIL are calculated only once prior to sewer flow routing. INFIL is called from Subroutine TRANS by setting the variable, NINFIL equal to 1, thus signaling the computer to estimate infiltration. In fact, however, the user has most of the responsibility for infiltration estimation, optionally using techniques described below. The program does little more than apportion it properly.
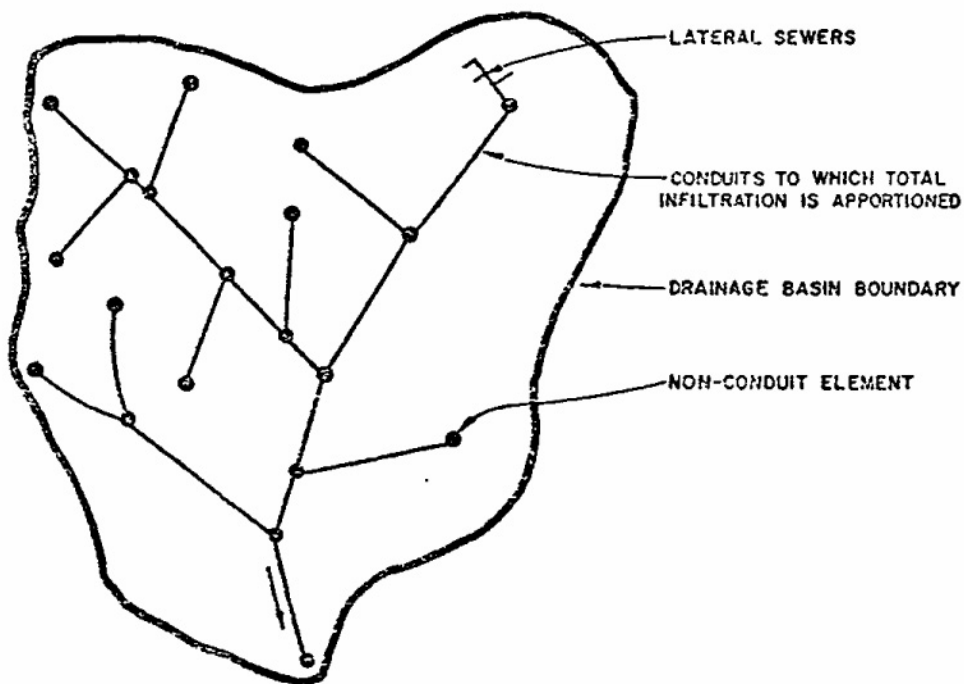
Figure 6-6. Typical drainage basin in which infiltration is to be estimated.

For the purposes of analysis, infiltration is classified into four categories, i.e., miscellaneous sources causing a base dry weather inflow, frozen residual moisture, antecedent precipitation, and high groundwater. The cumulative effects of the first three sources can be seen in Figure 6-7 which excludes surface runoff. Figure 6-7 shows total infiltration QINF as the sum of dry weather infiltration DINFIL, wet weather infiltration RINFIL, and melting residual ice and frost infiltration SINFIL. However, in cases where the groundwater table rises about the sewer invert, it is assumed that groundwater inflow GINFIL alone will be the dominant source of infiltration. Thus, infiltration is defined as:
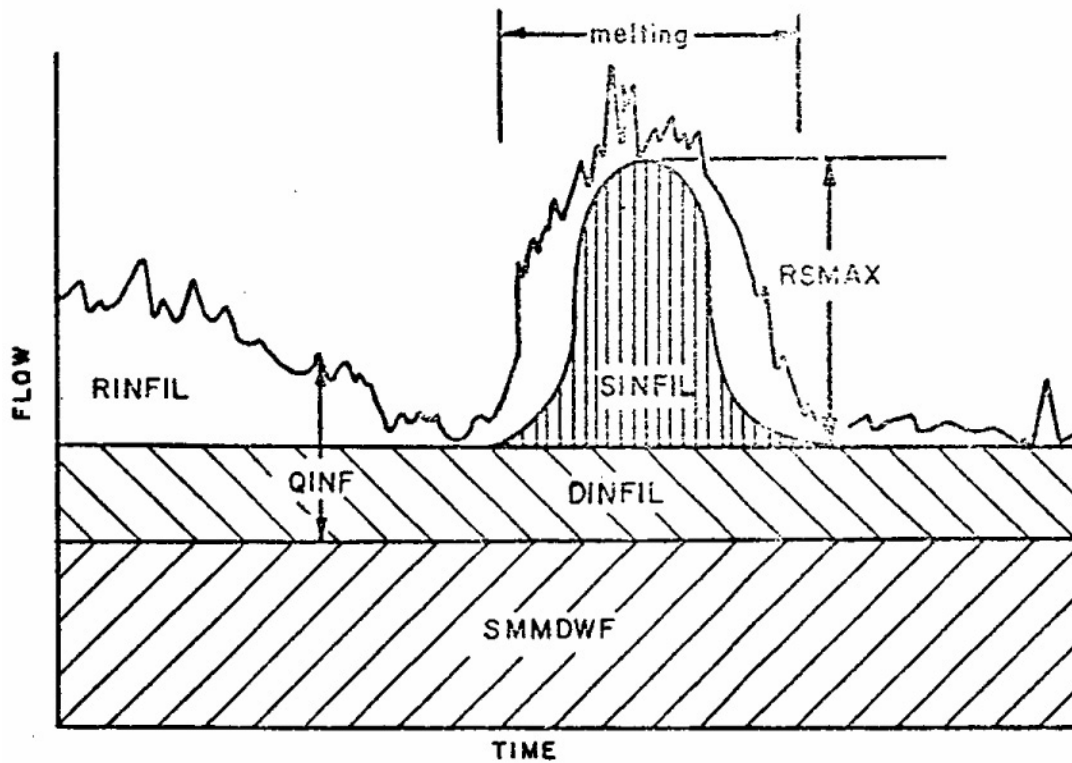
QINF	=	Total infiltration
DINFIL	=	Dry weather infiltration
RINFIL	=	Wet weather infiltration
SINFIL	=	Melting residual ice and snow infiltration
RSMAX	=	Residual moisture peak contribution
SMMDWF	=	Accounted for sewage

Figure 6-7. Components of infiltration

$$DINFIL + RINFIL + SINFIL$$

$$QINF = \qquad \text{or} \qquad\qquad\qquad\qquad\qquad\qquad\qquad (6\text{-}5)$$

$$GINFIL \text{ for high groundwater table}$$

Throughout the procedure for determining input variables, observations and estimates based upon local data are given preference over generalized estimates for infiltration described below. Thus, the hierarchy for basing estimates should be:
1) Use historical data for the study area under consideration.
2) Use historical data for a nearby study area and adjust results accordingly.
3) Use estimates of local professionals.
4) Use generalized estimates based upon country-wide observations.

Infiltration - inflow studies (e.g., EPA, 1977) have been performed in many cities and should provide much of the needed data.

Dry Weather Infiltration (DINFIL)

If the study area under consideration has been gaged, base dry-weather infiltration can be taken by inspection from the flow data. In the absence of flow data, an estimate of the unit infiltration rate XLOCAL (gpm/inch-diameter per mile) for dry weather must be obtained from local professionals. From data in this form, equation 6-6 can then be used to determine DINFIL (gpm):

$$DINFIL = XLOCAL \diamondsuit DIAM \diamondsuit PLEN$$

where

DIAM = average sewer diameter, in., and
PLEN = pipe length, mi.

Values of XLOCAL range from 250 to 600 gpm/in.-diameter per day (ASCE-WPCF, 1969) and may be even higher for laterals with many stubs and wyes. The importance of local data cannot be over-emphasized.

Residual Melting Ice and Frost Infiltration (SINFIL)

SINFIL arises from residual precipitation such as snow as it melts following cold periods. Published data (American Society of Heating and Air Conditioning Engineers) in the form of monthly degree days (sum of deviations below 65°F) provide an excellent index as to the significance of SINFIL. Average monthly degree-days for cities in the United States are reproduced in Appendix VIII. The onset and duration of melting can be estimated by noting the degree days NDD above and immediately below a value of 750. Refer to Figure 6-8 for the following description.

Figure 6-8. Prescribed melting period.
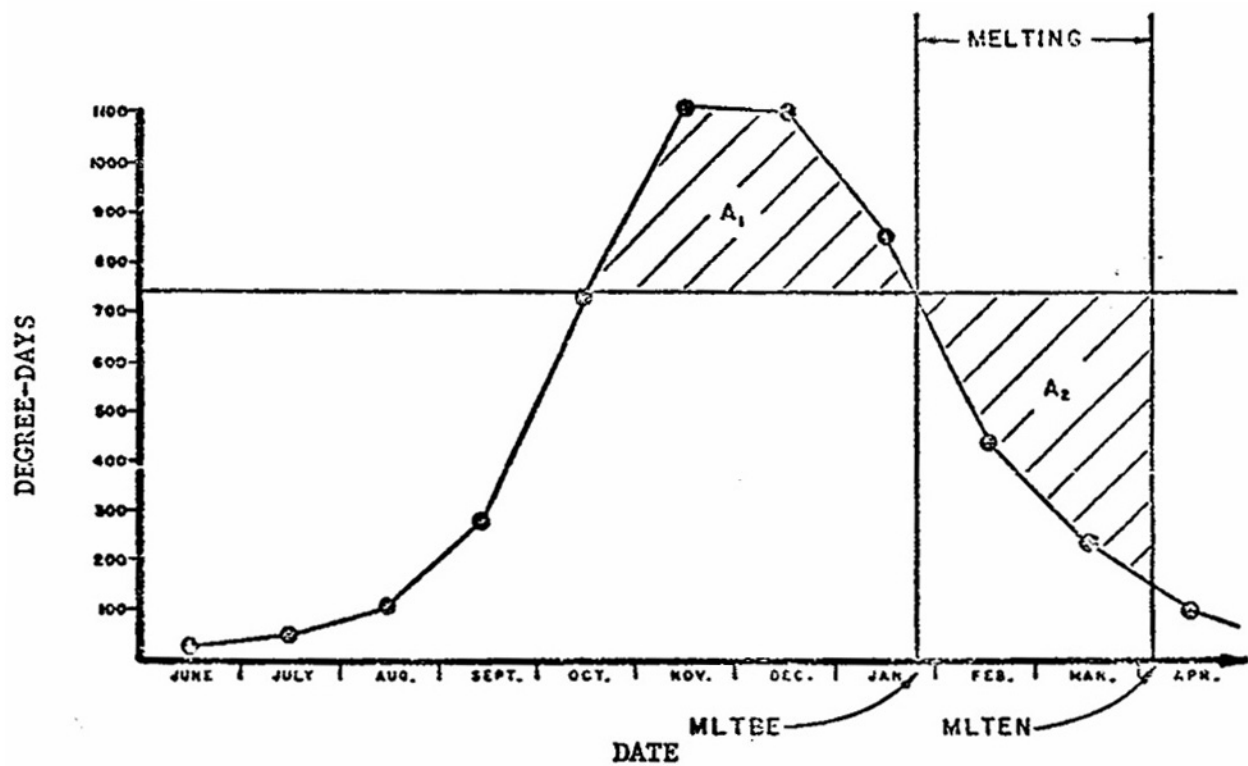
MLTBE	=	Day on which melting period begins
MLTEN	=	Day on which melting period ends

Within subroutine INFIL, the beginning of melting, MLTBE, is taken as the day on which NDD drops below 750. Next, MLTEN is determined so that A_1 equals A_2. In the absence of evidence to the contrary, it is assumed that the melting rate is sinusoidal. The maximum contribution RSMAX from residual moisture can be determined from previous gaging of the study area or local estimates. In either case, SINFIL is determined within the program by the following equation:

$$SINFIL = \begin{cases} RSMAX \cdot \sin[180 \cdot (NDYUD - MLTBE)/(MLTEN - MLTBE)] \\ 0.0 \text{ if } NDYUD \text{ is not melting period or if } NDD \text{ never exceeds } 750. \end{cases} \quad (6\text{-}7)$$

where

NDYUD	=	day on which infiltration estimate is desired,
RSMAX	=	residual moisture peak contribution, gpm,
MLTBE	=	beginning of melting period, day, and
MLTEN	=	end of melting period, day.

Note that RSMAX is a required input parameter, in addition to degree day information.

Antecedent Precipitation (RINFIL)

RINFIL depends upon antecedent precipitation occurring within the nine day period prior to an estimate. If antecedent rainfall is unavailable or less than about 0.25 in. (6.4 mm), the RINFIL contribution to QINFIL is usually small. For larger antecedent rainfall contributions, regression techniques offer one method of estimating RINFIL. For example, during development of the SWMM infiltration routine, available rainfall and infiltration data were examined (Metcalf and Eddy et al., 1971a). For three areas in which sewer flow data were not affected by melting, RINFIL was found to satisfy the following linear relationship:

$$RINFIL = ALF + ALF0 \diamondsuit RN0 + ALF1 \diamondsuit RN1 + \ldots + ALF9 \diamondsuit RN9$$

where

RINFIL	=	SWFLOW - DINFIL - SMMDWF, gpm,
ALFN	=	coefficient to rainfall for N days prior to estimate, gpm/in.,
RNn	=	precipitation on n days prior to estimate, in.,
SWFLOW	=	daily average sewer flow excluding surface runoff, gpm, and
SMMDWF	=	otherwise accounted for sewage flow, gpm.

To determine the coefficients in equation 6-8, a multiple linear regression should be run on existing flow and rainfall data. For comparative purposes, the results of regression analyses for study areas in three selected cities (Lentz, 1963; Metcalf and Eddy et al., 1971a) are given in Table 6-5.

High Groundwater Table (GINFIL)

For locations and times of the year that cause the groundwater table to be above the sewer invert, groundwater infiltration GINFIL supersedes contributions from DINFIL, RINFIL, and SINFIL. GINFIN can be determined from historical sewer flow data by inspection or regression analysis. For example, a regression analysis could involve determination of the BETA coefficients in Equation 6-9, or an alternative formulation could be investigated.

Table 6-5. RINFIL Equations for Three Study Areas

Study Area	Equation
Bradenton, Florida	RINFIL = 4.1 + 2.9RN0 + 17.5RN1 + 15.0RN2 + 12.8RN3 + 13.0RN4 + 10.4RN5 + 13.2RN6 + 10.1RN7 + 11.8RN8 + 9.5RN9
Baltimore, Maryland	RINFIL = 2.4 + 11.3RN0 + 11.6RN1 + 5.5RN2 + 6.4RN3 + 4.8RN4 + 3.6RN5 + 1.0RN6 + 1.5RN7 + 1.4RN8 + 1.8RN9
Springfield, Missouri	RINFIL = 2.0 + 18.3RN0 + 13.9RN1 + 8.9RN2 + 5.5RN3 + 6.7RN4 + 16.RN5 + 5.2RN6 + 4.6RN7 + 4.4RN8 + 1.3RN9

$$\text{GINFIL} = \text{BETA} + \text{BETA1} \diamond \text{GWHD} + \text{BETA2} \diamond \text{GWHD}^2 + \text{BETA3} \diamond \text{GWHD}^{0.5}$$
(6-9)

where

GWHD = groundwater table elevation above sewer invert, ft, and

BETAn = coefficient for term n.

Apportionment of Infiltration

Once an estimate of the total local infiltration QINF has been obtained, this flow must be apportioned throughout the designated study area. The criterion chosen for apportionment is an opportunity factor OPINF which represents the relative number and length of openings susceptible to infiltration. Pipe joints constitute the primary avenue for entry of infiltration (Geyer and Lentz, 1963). The number and length of joints is assumed to be proportional to the relative surface area of each conduit. For each, an equivalent circular pipe diameter will be proportional to the square root of its known cross-sectional area, ft^2. Then the fraction of total infiltration ("opportunity" for infiltration) allocated to each conduit, OPINF, is:

$$\text{OPINF} = \frac{A_f \cdot \text{DIST}}{\sum A_f \cdot \text{DIST}}$$
(6-10)

where

A$_f$ = cross sectional area of conduit, ft^2, and

DIST = conduit length, ft.

The summation in the denominator is over all conduits. Open trapezoidal channels are treated the same as all others. The apportioned infiltration enters the system at the non-conduit element immediately upstream of the conduit.

237

This procedure allocates the most infiltration to the largest and longest conduits. Should local information dictate otherwise, infiltration may be apportioned and entered at appropriate manholes in data group E1.

Infiltration developed using subroutine INFIL is held constant in time. Should hourly or daily corrections be desired, infiltration can be incorporated into dry-weather flow or entered as hydrographs using data group R1.

Quality of Infiltration

Although infiltration is often assumed to be "clean" due to its origin in the soil layers, in-conduit measurements usually indicate non-zero levels of most parameters. These concentrations may be entered in data group K1.

Data Needs
Hydrologic Data

Concurrent historical rainfall, water table, and sewer flow data of several weeks' duration are needed to completely describe infiltration. In addition, rainfall for several days prior to the flow estimate is required for use in a regression equation for RINFIL. Of course, such data would be required for many different storms for development of such an equation.

Ideally, the rainfall record would be from a raingage which is located near the center of the study area and which records daily rainfall in inches. If more than one raingage is located within the study area, daily measurements from all gages should be averaged. Missing data (e.g., from a malfunctioning gauge) or a total absence of measurements due to no gauging within the study area can sometimes be overcome with measurements taken from a raingage located within a few miles. If National Weather Service (NWS) climatological data recorded at the nearest airport or federal installation are not available, contact the National Climatic Data Center (Asheville, NC) for assistance.

Should some other form of precipitation, e.g., snowfall, be encountered, it will be necessary to convert this to equivalent rainfall. If estimates are unavailable from the NWS, the ratio of ten inches of snow to one inch of rain may be used.

Water table data should also be obtained from gauging within the study area. However, shallow-well data from the U.S. Geological Survey or state geological office can be used to supplement missing data. Water table elevations are not required if they are below the sewer inverts for the day on which QINF is to be estimated.

Sewer Data

Sewer flow data for regression analysis should be taken from a gage located at the downstream point within the study area. Upstream gaging may sometimes be used to estimate flows at the downstream point by simply adjusting flows based upon respective surface area. Physical sewer data (e.g., lengths, diameters) are taken from prior input used within TRANS to route sewer flow.

Summary of Infiltration Procedures
Input

Effective use of the Infiltration Model requires estimates of its component flows, namely:

DINFIL   =   dry weather infiltration,

RINFIL   =   wet weather infiltration,
SINFIL   =   melting residual ice and snow, and
GINFIL   =   groundwater infiltration.

Step 1. Determine Groundwater Condition

If the groundwater table is predominantly above the sewer invert, all infiltration is attributed to this source (GINFIL). In this case, an estimate of the total infiltration is made directly (in cfs for the total drainage basin) and read in data line K1. This data line followed by a blank data line (data line K2) would complete the infiltration data input. If the groundwater table is not predominantly above the sewer invert, proceed to Step 2.

Step 2. Build-Up Infiltration from Base Estimates

From measurements, historical data, or judgment, provide estimates of DINFIL and RINFIL. In this case, GINFIL must be set equal to 0.0. Finally, if needed, provide the peak residual moisture (RSMAX) and the 12 monthly degree-day totals taken from Appendix VII or a local source.

Dry Weather Flow Model
Methodology

Subroutine FILTH serves as an option to estimate average sewage flow and quality from residential, commercial, and industrial urban areas. FILTH estimates sewage inputs at discrete locations along the trunk sewers of any specified urban drainage basin. These estimates are calculated from data describing drainage basin subsections (subcatchments and subareas) under which the trunk sewer passes. In this routine, dry-weather flow quantity and quality are developed from regression equations, as explained in the documentation (Metcalf and Eddy et al., 1971a). The estimates are for three specific quality parameters; BOD_5, suspended solids (SS) and total coliforms. Thus, if any different parameters are to be simulated, FILTH cannot be used. However, if a fourth parameter is to be routed in addition to BOD_5, SS and total coliforms, FILTH can be used to provide estimates for the first three but not the additional one. Also bear in mind that a constant base flow for any parameter may be input at manholes in data group E1.

When FILTH is not used, DWF estimates may be input at desired manholes, as discussed previously. In fact, this option may be routinely used in place of FILTH whenever reasonable estimates are available for instream DWF quantity and quality, thus avoiding collection of the many demographic data required for estimates by FILTH.

An example of a hypothetical sewer system and input situation for FILTH is given in Figure 6-9. To avoid confusion with Runoff Block subcatchments, all drainage basin subdivisions will be referred to as subareas in the following discussion. As shown in the figure, an input manhole near the center of each subarea is assumed to accept all sewage flow from that subarea. Criteria for establishing subarea boundaries and input locations are discussed later in the text.

In the context of SWMM, FILTH calculates daily sewage flow (cfs) and characteristics (BOD_5, SS, and total coliforms) averaged over the entire year for each subarea. FILTH is called from Subroutine TRANS by setting the parameter NFILTH equal to one. Flow and quality characteristics estimates and corresponding manhole input numbers are then returned to TRANS where the estimates undergo adjustment depending upon the day of the week and hour of the day during which simulation is proceeding.

The subroutine may be omitted when modeling separate storm sewers unless it is desired to generate a base flow with DWF characteristics. FILTH is designed to handle an unrestricted number of inlet areas and individual process flow contributors. As a safeguard against faulty data, however, a program interrupt is provided if the combined number exceeds 200, which is a limit set by the Transport Model.

SEWER ELEMENT NUMBERS

MANHOLE

SUBCATCHMENT OR SUBAREA NUMBER

INPUT MANHOLES

CONDUITS

SUBAREA BOUNDARIES

SUBCATCHMENT BOUNDARIES
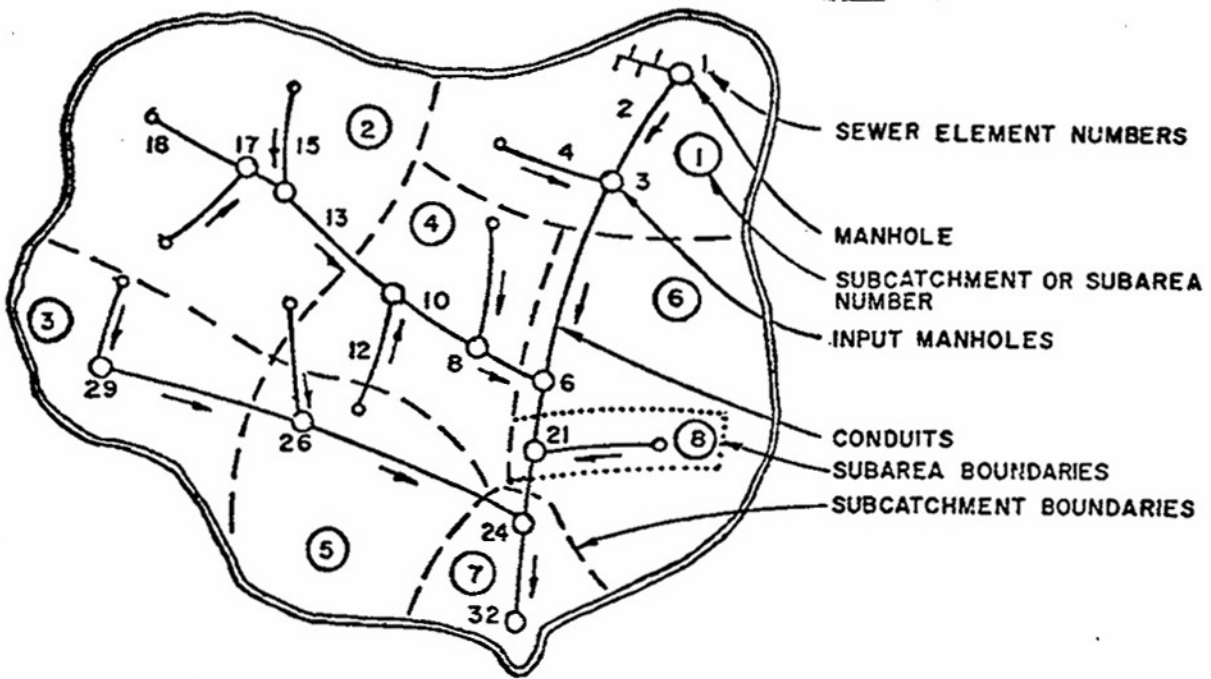
Sewer and Subcatchment Data

1. Manhole 32 is the most downstream point.
2. Subcatchments 1, 2, 3, and 4 are single-family residential areas, each 100 acres in size and each with water metering.
3. Subcatchments 5 and 7 are 220-acre industrial areas.
4. Subarea 6 is a 250-acre park.
5. Subarea 8 is a 50-acre commercial area.

   Subareas 6 and 8 constitute a subcatchment draining to input manhole number 21.

Resulting Data

8 sewage estimates
KTNUM, Total subcatchments and subareas in drainage basin = 8
TOTA, total acres in drainage basin = 1,140

KNUM, subcatchment or subarea	INPUT, input manhole number	KLAND, land use category	ASUB, acres in subcatchment or subarea
1	3	1	100
2	17	1	100
3	29	1	100
4	8	1	100
5	26	4	220
6	21	5	250
7	24	4	220
8	21	3	50

Figure 6-9. Determination of subcatchment and identification to estimate sewage at 8 points.

241

Quantity Estimates

Data Categories

Three data categories are used to estimate sewage flow: (1) drainage basin data, (2) subarea data, and (3) decision and adjustment parameters.

Study area data are TOTA, KTNUM and ADWF. KTNUM denotes the number of subareas into which a drainage basin, having a surface area TOTA (acres), is being divided. ADWF, which is optional depending upon its availability, gives the average sewage flow (cfs) originating from the entire drainage basin (e.g., average flow data from a treatment plant serving the study area). When it is included, the predicted basin flow will be adjusted to match this value.

Subarea data requirements consist of several options depending upon availability and choice of input. Discussion later in the text will assist in data tabulation by noting the order of preference where options exist. Subarea data can be broken into three categories as follows: (1) identification parameters, (2) flow data, and (3) estimating data.

Identification Parameters

Identification parameters are KNUM, INPUT, AND KLAND. KNUM identifies each subarea by a number less than or equal to KTNUM. For each of the KTNUM subareas, INPUT indicates the number of the manhole into which DWF is assumed to enter. Land use within each subarea which approximately corresponds to zoning classification is categorized according to Table 6-6. KLAND serves as an important factor in deciding subarea locations and sizes. Figure 6-9 will assist in describing how the above data are determined and tabulated.

Table 6-6. Land Use Classifications

KLAND	Land Use
1	Single-family residential
2	Multi-family residential
3	Commercial
4	Industrial
5	Park and open area

Flow Data

Flow data are optional inputs that eliminate the need for using predictive equations. Two possible types of flow data are average sewage flow measurements, SEWAGE, and metered water use, WATER. Commercial or industrial sewage flow or water use measurements should be input using the variable SAQPF. Flows from commercial and industrial establishments located in residential or open subareas may be included using SAQPF, also. Metering at lift stations and other flow control structures within the study area is occasionally available and should be used whenever possible. Metered water use offers a more available source of subarea flow data. Unfortunately, considerable effort in locating, tabulating, and averaging these data is often required.

Estimating Data

For each subarea where SEWAGE or WATER measurements are not available estimated water use must be used as an estimate of sewage flow. In the case of a factory or commercial establishment, estimates can be made by multiplying the number of employees by an established coefficient (gpd per employee). In the case of a large factory or commercial establishment, one subarea may be established with estimated water use tabulated as SAQPF for that subarea. On the other hand, estimates of water use for established non-residential areas (e.g., industrial parks or shopping centers) may be summed and tabulated as SAQPF for one large subarea. A list of the above mentioned coefficients is given in Appendix VIII.

In the case of residential areas, estimating data for each subarea are METHOD, PRICE, ASUB, POPDEN, DWLINGS, FAMILY, and VALUE. Default values and definitions of each of these are given in the description of input data.

Decision and Adjustment Parameters

These parameters consist of DVDWF, HVDWF, KDAY, CPI, and CCCI. DVDWF and HVDWF are daily and hourly correction factors, respectively, for DWF. DVDWF is comprised of seven numbers that are ratios of daily average sewage flows to weekly average flow. Likewise, HVDWF is comprised of 24 numbers that are ratios of hourly average sewage flows to daily average flow. Both groups of numbers may be derived from observed flow variation patterns throughout the country (e.g., Tucker, 1967; Portland Clement Association, 1968). Their use is to correct measured or estimated average sewage flow to more accurate estimates depending upon the day and hour. Typical sewage flow variations are shown in Figures 6-10 and 6-11. These flow patterns are only examples; locally observed patterns more accurately describe local variations and should be used when available.

KDAY denotes the day of the week at which simulation is to begin. As the simulation proceeds, this value is continually updated. By using the current day and hour, the appropriate values of DVDWF and HVDWF can be multiplied by average flow to determine the correct value. KDAY ranges from 1 to 7 with Sunday being day number 1.

Two cost indices are employed to adjust current house valuations and water prices to appropriate 1960 values and 1963 prices, respectively. This is done because estimating equations within FILTH are based upon 1960 values and 1963 prices. CPI, consumer price index, has been chosen to adjust water price by multiplying water price by 1960 CPI divided by the current CPI. CCCI, composite construction cost index, has been chosen to adjust house valuations similarly. Both indices can be found in most libraries in journals on economic affairs (e.g., U.S. Dept. of Commerce, Survey of Current Business and Statistical Abstracts of the United States).

Quality Estimates

The purpose of the DWF quality computation is to apportion waste characteristics (such as would be measured at a sewage treatment plant before treatment) among the various subareas in the drainage basin under study, or in the event no measured data are available, to estimate and apportion usable average values. The apportionment is based upon the flow distribution, land use, measured or estimated industrial flows, average family income, the use or absence of garbage grinders, and infiltration.

Daily and hourly correction factors for concentrations of BOD_5, SS and total coliforms are input in conjunction with those for flow variations. All are expressed as ratios of instantaneous to annual or daily averages.
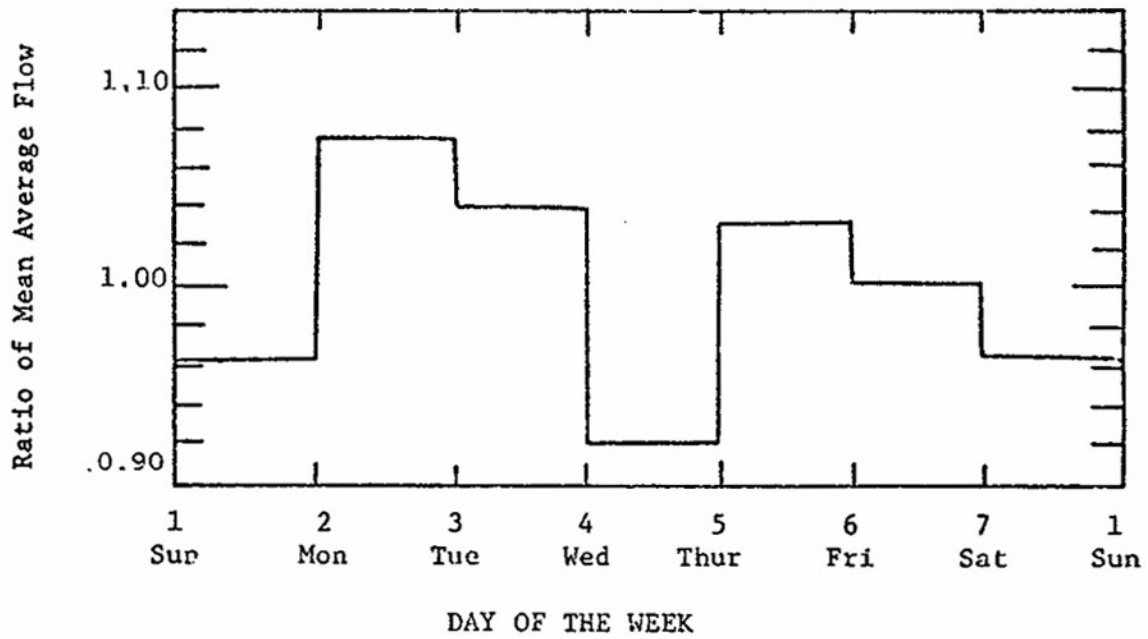
Figure 6-10. Representative daily flow variation.
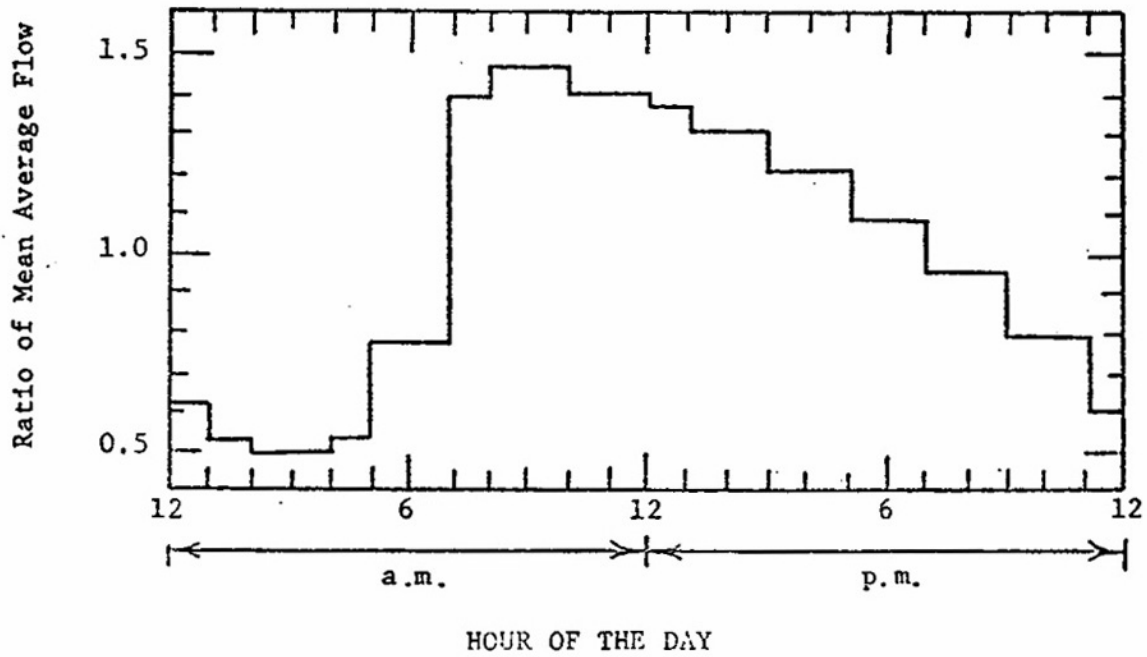
Figure 6-11. Representative hourly flow variation.

Data line N1 includes the total number of subareas and process flow sources to be processed along with the type case (whether the total DWF characteristics are known or to be estimated), the number of process flow contributors, the cost indices, and the total drainage basin population. Depending upon the instructions given, computations proceed along the Case 1 or Case 2 channel.

- Case 1 – In this instance, the total DWF quality characteristics are known at a point well downstream in the system. These characteristics may be obtained from treatment plant operating records (raw sewage) or by a direct sampling program. The average daily concentrations are read into the program for flow, BOD_5, SS, and total coliforms (data line O1). The total pounds per day of BOD_5 and SS and the total MPN per day of coliforms are then calculated. Then, infiltration and baseflow are subtracted from the average daily flow. Note that infiltration is computed in separate Subroutine INFIL. If it is not executed a default of zero will be assumed.

  Next, the known process flow contributions (data group P1) are summed and deducted from the daily totals, yielding a further corrected flow, C2DWF (cfs), and characteristics, C1BOD and C1SS (lb/day). This is the only use of the input from data group P1. Process flow information must be re-entered for each subarea, in data group Q1.

  Finally, corrections based on regression equations, are made for personal income variations, degree of commercial use, and garbage grinder status (data line O2). The DWF quantity does not change but the characteristics obtain new, average values, C2BOD and C2SS. Average concentrations of the residual flow, A1BOD, A1SS, and A1COLI are then computed.

- Case 2 – Here no direct measurements are available; thus, estimates must be made or default values will be assumed. A typical application of Case 2 would be in a situation where several catchments are to be modeled, yet funds will permit monitoring the DWF only in a single area. A1BOD, A1SS, and A1COLI would be computed via the Case 1 subroutine for the known area and the results could be transferred as Case 2 for the remaining catchments.

  Default values of A1BOD, A1SS, and A1COLI are 1300 lb/day-cfs (241 mg/l), 1420 lb/day-cfs (263 mg/l) and 6.2×10^7 MPN/100 ml. These values assume 85 gal/capita-day (322 l/capita-day) domestic wastewater flow and 0.02 lb/captia-day (0.09 kg/capita-day) for BOD_5, 0.22 lb/capita-day (0.1 kg/capita-day) for SS and 200 billion MPN/capita-day for total coliforms. All values assume average income families. The default value for ADWF assumes 100 gal/capita-day (376 l/capita-day) which includes an extra 15 gal/capita-day (57 l/capita-day) for infiltration or other sources.

Following estimation of basin totals, average daily flow and quality values are computed for each of the KTNUM subareas. Data are input in data group Q1 for estimation of water use and sewage quality as well as process flow for each subarea.

Dry weather flow quantity (DWF in cfs) is computed for each land use on the basis of the following priorities:

Priority	Method
1	Measured average sewage flow (SEWAGE ≠ 0.0).
2	Measured water use (WATER ≠ 0).
3	Regression equations, for single and multiple-family residential land use only.

The first two methods are really equivalent since DWF is simply equated to either SEWAGE or WATER, in this order, for all land uses. Regression equations are employed as a third choice for residential land uses. As explained in the documentation (Metcalf and Eddy et al., 1971a), DWF becomes a function of the number of dwelling units within the subarea (DWLNGS) and other parameters as listed below. DWLNGS is required for all regression equations and is computed on the following basis:

Priority	Method
1	Input on data line Q1
2	DWLNGS = POPDEN◇ASUB/FAMILY
3	Default to 10 units per acre.

DWF is then computed using DWLNGS and input parameters as listed below:

METHOD = 1		METHOD = 2
PRICE = 0	PRICE ≠0	
DWLNGS	DWLNGS	DWLNGS
VALUE	PRICE	FAMILY
	CPI	VALUE
	VALUE	

For each technique default values will be used where necessary. It may be inferred that parameters not used in a regression equation may be omitted from input. Note that VALUE is also used in each technique. It is adjusted to the 1960 Composite Construction Cost Index, CCCI, by

$$\text{VALUE} = \text{VALUE} ◇ 103/\text{CCCI} \tag{6-11}$$

Finally, the user is reminded that all inputs for the regression equations can be avoided if either SEWAGE or WATER is known.

For commercial, industrial or undeveloped land uses parameter SEWAGE or WATER is the only method used to input DWF, except that process flows are <u>added</u> to the value of DWF previously computed, for <u>all</u> land uses. Thus, they could constitute the only dry-weather flow source for non-residential land use.

Dry-weather flow quality starts with the average BOD_5 and SS concentrations (A1BOD and A1SS) previously computed for the entire subarea. These are used for the concentrations of non-process flows for all subareas, with two exceptions. First, for commercial and industrial areas, the average concentrations are multiplied by 0.9. Second, the strengths of residential flows are adjusted according to average family income, XINCOM, and percent garbage grinders, PCGG, as explained in the documentation.

The process flow load (i.e., flow times concentration) is then added to the loads just computed, for all land uses. For non-residential land use process flows could constitute the only quality loads.

Finally, for all subareas, total coliforms are computed solely on the basis of population using the average concentration, A1COLI, computed earlier along with the total basin populations, POPULA, (data group N1) and subarea populations computed from POPDEN (data group Q1). Thus, there will be a subarea contribution of total coliforms only if POPDEN = 0.

For each of the KTNUM subareas, subtotals (cumulative up to this subarea) of computed flows and quality will be printed for each subarea if MSUPT = 1. Otherwise, only basin totals will be printed. If measured basin averages have been input in group O1 (KASE = 1) all subarea loads are adjusted a constant ratio such that the flow and concentrations computed from the data of group Q1 will agree with the input averages.

Summary of Dry Weather Flow Requirements
Step 1. Establishing Subareas

Establishment of the subareas constitutes the initial step in applying subroutine FILTH. Both detail of input data and assumptions made in developing FILTH imply constraints on the type, size, and number of subareas. However, most important in subarea establishment is the type of estimating data available and the maintenance of homogenous land use.

Subareas should be located and sized to utilize existing sewer flow measurements taken within the drainage basin. These measurements should be recent and of sufficient duration to provide a current average sewage flow value for the period of time during which simulation is to proceed. Measured daily and hourly flow variation should be used in lieu of generalized values described earlier in the text. A gaging site with less than 200 ac (81 ha) contributing flow often provides a convenient data input situation. A subarea should be established upstream from the gage with average sewage flow tabulated as SEWAGE for that subarea. It is convenient, though not necessary, for the subareas to correspond to subcatchments in the Runoff Block.

If metered water use is to be used to estimate sewage flow, subareas should be located to coincide with meter reading zones or other zones used by the water department that simplify data takeoff. Since water use would be used to estimate sewage flow, average winter readings should be used to minimize the effects of lawn sprinkling and other summer uses.

If neither gaging nor metered water use are input, sewage characteristics must be estimated. Subareas should then be established to yield appropriate input data for the residential estimating equations in FILTH. Zero sewage flow is assumed from commercial, industrial, and parkland subareas for which SEWAGE and WATER are zero and measurements of SAQPF are not given. Since KLAND and VALUE are the significant variables in estimating subarea sewage flow, subareas should be located and sized to include land with uniform land use and property valuation. To utilize existing census data, subarea boundaries should be made to coincide with census tract boundaries.

Criteria for establishing subareas are listed in the following summary:

1) Subareas in general should:
   a. be of homogenous land use;
   b. be less than or equal to 159 in number; and
   c. conform to the branched pipe network.
2) Subareas should be established to employ any existing sewer flow measurements.
3) Subareas for which metered water use is used to estimate sewage flow should be compatible with meter reading zones.
4) Residential subareas for which estimated water use is used to estimate sewage flow should:
   a. be uniform with respect to land use;
   b. be uniform with respect to dwelling unit valuation; and
   c. coincide with census tracts.

Step 2. Collection of Data

Other than the establishment of measured data described earlier, the primary data source is the Bureau of Census for census tract information. This source provides readily available data on population distribution, family income, and the number and relative age of dwelling units. City records, aerial photographs, and on-site inspection may be necessary to define land use activities, process flow, and dwelling density variations within tracts.

Step 3. Data Tabulation

Once subareas have been established, several alternatives exist regarding data tabulation. An identification number KNUM should be given to each subarea prior to data takeoff. However, once KNUMs have been established, corresponding INPUT manhole numbers are selected from a previously numbered schematic diagram of the trunk sewer. This numbered schematic serves as the mechanism to coordinate runoff, infiltration, and sewage inputs. Refer to the earlier discussion for additional information about the numbered schematic. If water use estimates are necessary, land use should be determined from city zoning maps and the previously tabulated values for KLAND.

ADWF should be tabulated as average drainage basin sewage flow. As the ADWF, SEWAGE should be averaged from flow data for the appropriate month, season, or year. ADWF, SAQPF, or SEWAGE may be obtained from routine or specific gaging programs done by the city, consulting engineers, or other agencies. SAQPF may be estimated for commercial and industrial areas using water use coefficients (Appendix VIII). Also, SAQPF and WATER may be determined for all land use categories from water meter records.

Initialization

Following execution of subroutines INFIL and/or FILTH, flows and concentrations will be initialized to baseflow values simply by summing flows and loads at all junctions (non-conduits) in subroutine INITAL. Baseflow can thus originate from three sources: input at manholes, infiltration (Subroutine INFIL), and/or dry-weather flow (Subroutine FILTH). Inflows from FILTH are always subject to the hourly and daily adjustment factors; inflows from INFIL and manholes are not.

In addition, the buildup of settled pollutant fractions (if simulated) in the sewer system is estimated using Subroutine DWLOAD. For the particle size distribution and specific gravity discussed earlier, daily "solids" deposition is computed for DWDAYS dry-weather days (data group B2) prior to the simulation. The initial pounds of deposition are printed for each conduit. This material is then eligible for erosion during the simulation (computed in Subroutine QUAL). Thus, if flows increase over their initial values, (as expected during a storm) a "first flush" will be provided.

Output

Input hydrographs and pollutographs will be printed for nonconduit elements selected in data group J1 and ouput hydrographs and pollutographs printed for nonconduit elements selected in group J2. A print interval INTPRT (group B1) is used. For example, if INTPRT = 2, values for every other time step will be printed. If INTPRT = 0, only the average, standard deviation, and total volume of flow (or mass of pollutant) will be printed. Graphical output is useful and is accomplished by saving hydrographs and pollutographs for desired locations using data group H1, followed by application of the Graph Block. Any nonconduit location in the system may be saved for graphing, not just the most downstream points.

A continuity check is printed breaking down the sources of inflow among external hydrographs, infiltration, and dry-weather flow. A continuity check is also performed for each quality constituent that includes external inputs, dry-weather flow and infiltration, and scour and deposition. If surcharging has occurred a summary table is printed listing location, duration and ratio of required inflow to pipe capacity. Finally, if the hydraulic design routine is used to re-size conduits (option NDESN on group B3), a final table of dimensions for all conduits is printed.

Table 6-7. Transport Block Input Data

There have been many changes made to the input format of SWMM. Following is a short list of the major changes along with explanations and guidelines.

1. Free format input. Input is no longer restricted to fixed columns. Free format has the requirement, however, that at least one space separate each data field. Free format input also has the following strictures on real, integer, and character data.
    a. No decimal points are allowed in integer fields. A variable is integer if it has a 0 in the default column. A variable is real if it has a 0.0 in the default column.
    b. Character data must be enclosed by single quotation marks, including both of the two title lines.
2. Data group identifiers are a requirement and <u>must</u> be entered in columns 1 and 2. These aid the program in line and input error identification and are an aid to the SWMM user. Also blank lines no longer are required to signal the end of sets of data group lines; the data group identifiers are used to identify one data group from another.
3. The data lines may be up to 230 columns long.
4. Input lines can wrap around. For example, a line that requires 10 numbers may have 6 on the first line and 4 on the second line. The FORTRAN READ statement will continue reading until it finds 10 numbers, e.g.,

        Z1 1 2 3 4 5 6
              7 8 9 10

    Notice that the line identifier is <u>not</u> used on the second line.
5. An entry must be made for <u>every</u> parameter in a data group, even if it is not used or zero and even if it is the last required field on a line. Trailing blanks are not assumed to be zero. Rather, the program will continue to search on subsequent lines for the "last" required parameter. Zeros can be used to enter and "mark" unused parameters on a line. This requirement also applies to character data. A set of quotes must be found for each character entry field. For instance, if the two run title lines (data group A1) are to consist of one line followed by a blank line, the entry would be:

        A1 'This is line 1.'
        A1 ''

6. See Section 2 for use of comment lines (indicated by an asterisk in column 1) and additional information.

Variable	Description	Default
	Two Title Lines	
A1	Group identifier	None
TITLE	Title, two lines with heading to be printed on output[1] (max., 80 columns each).	None
	First Control Data Group	
B1	Group identifier	None
NDT	Total number of time-steps, no limit.[3]	0
NINPUT	Total number of non-conduit elements into which there will be data input of hydrographs and pollutographs in Group R1 (maximum = 80).[5]	0
NNYN	Total number of non-conduit elements at which input hydrographs and pollutographs are to be printed out (maximum = 80, minimum = 0).	0
NNPE	Total number of non-conduit elements at which routed hydrographs and pollutographs are to be printed out (maximum = 80, minimum = 0).[7]	0
NOUTS	Total number of non-conduit elements at which flow is to be transferred to a subsequent block by disk file (maximum = 100).[8]	0

Table 6-7. Continued

Variable	Description	Default
NPRINT	Control parameter for program-generated error messages (e.g., non-convergence) occurring in the execution of the flow routing scheme. These errors do not normally affect the program execution.[2] = 0, messages suppressed = 1, messages printed	0
NPOLL	Total number of pollutants being routed (maximum = 4, minimum = 0). When NPOLL = 0, program will route flows only and all quality operations will be bypassed.	0
NITER	Total number of iterations to be used in routing subroutine (4 recommended).	4
IDATEZ	Starting date of storm, year-month-date e.g., July 20, 1979 = 790720. Superseded by value on interface file from previous block if accessed.	000000
METRIC	Metric input/output. = 0, U.S. customary units = 1, Metric units, indicated in brackets [ ] among input variables.	0
INTPRT	Print interval for input and output elements. Use 1 for printing at each time step. Value of zero will result in printing of total loads and moments only.	0
Second Control Data Group		
B2	Group identifier	None
DT	Size of time-step for computation, sec.[3,4]	0
EPSIL	Allowable error for convergence of iterative methods in routing routine (0.0001 recommended).	0.0001
DWDAYS	Total number of days (dry weather days) prior to simulation during which solids were not flushed from the sewers.	0
TZERO	Starting time of day of storm, hours and fraction, e.g., 5:30 p.m. is 17.5. Superseded by value on interface file from previous block, if accessed.	0
GNU	Kinematic viscosity of water, ft^2/sec [cm^2/sec]. Required only if SPG > 1.0 for any of pollutants in group F1.	10^{-5} [10^{-2}]
TRIBA	Total catchment area, ac [ha]. Superseded by value on interface file from previous block, if accessed.	0.0
Third Control Data Group		
B3	Group identifier	None
NCNTRL	Control parameter specifying means to be used in transferring inlet hydrographs. = 0, Input from a preceding block using interface file JIN (line input, group R1, optional).[5] = 1, No input from interface file. Line input (data group R1) only (optional).	0
NINFIL	Control parameter in estimating sewer infiltration inflows.[6] = 0, Infiltration not estimated (INFIL not called and corresponding data omitted). = 1, Infiltration to be estimated (subroutine INFIL called).	0

Table 6-7. Continued

Variable	Description	Default
NFILTH	Control parameter in estimating dry-weather sewage inflow.[6] If used with quality simulation, the first three pollutants must be BOD5, SS and total coliforms. = 0, Sewage inflows not estimated (subroutine FILTH not called and corresponding data omitted). = 1, Sewage inflows to be estimated (subroutine FILTH called).	0
NDESN	Control parameter for hydraulic design routine (resizing of undersized conduits). = 0, Hydraulic design routine is not called. = 1, Hydraulic design routine is to be called.	0
	Flow Routing Data for New Shapes	
C1	Group identifier	None
NKLASS	Number of user-supplied sewer cross-sectional shapes, in addition to the 16 program-supplied, for which element routing parameters are to follow (maximum value = 2). These will become types 17 and 18.	0
KPRINT	Control parameter for printing flow routing parameters for all shapes (about 500 lines). = 0, Suppress printing. = 1, Allow printing (for all shapes, program-supplied and additional)	0
	OMIT GROUPS D1 TO D9 IF NKLASS = 0	
D1	Group identifier	None
NAME(I,17)	16 – letter name of shape 1	'None'
NAME(I,18)	16 – letter name of shape 2	'None'
	Number of values of DNORM to be input (max = 51, min = 2)	
D2	Group identifier	None
NN(17)	Number of DNORM values for shape 1.	0
NN(18)	Number of DNORM values for shape 2.	0
	Number of values of QNORM to be input (max = 51, min = 2)	
D3	Group identifier	None
MM(17)	Number of QNORM values for shape 1.	0
MM(18)	Number of QNORM values for shape 2.	0
	Value of A/A_f[9] corresponding to the maximum Q/Q_f value of each shape	
D4	Group identifier	None
ALFMAX(17)	A/A_f value for shape 1	0
ALFMAX(18)	A/A_f value for shape 2	0
	Maximum Q/Q_f[11] value for each shape	
D5	Group identifier	None
PSIMAX(17)	Maximum Q/Q_f value for shape 1	0
SIMAX(18)	Maximum Q/Q_f value for shape 2	0
	Factor used to determine full flow area for each shape, i.e., for use in AFULL = AFACT(GEOM1)[9]	
D6	Group identifier	None
AFACT(17)	Factor for shape 1	0
AFACT(18)	Factor for shape 2	0

Table 6-7. Continued

Variable	Description	Default
	Factor used to determine full flow hydraulic radius for each shape, i.e., for use in equation RADH = RFACT(GEOM1)	
D7	Group identifier	None
RFACT(17)	Factor for shape 1	0
RFACT(18)	Factor for shape 2	0
	REPEAT GROUP D8 FOR EACH ADDED SHAPE	
	Input of tabular data (depth of flow, y, divided by total depth of conduit, $y_f = y/y_f$) for each added shape corresponding to the NN-1 equal divisions of A/A_f of the conduit as given by NN in Group D2.[10] 8 values per line.	
D8	Group identifier	None
DNORM(I,1)	First value for y/y_f for shape 1	0
DNORM(I,2)	Second value for y/y_f for shape 1	0
■	■	
DNORM(I,NN(I))	Last value of y/y_f for shape 1	0
	(Total of NN(17)/8 + NN(18)/8 data groups)	
	REPEAT GROUP D9 FOR EACH ADDED SHAPE	
	Input of tabular data (flow rate, Q, divided by the flow rate of the conduit running full, Q_f, $= Q/Q_f$) for each added shape corresponding to the MM-1 equal divisions of A/A_f of the conduit as given by MM in group D3.[11] 8 values per line.	
D9	Group identifier	None
QNORM(I,1)	First value of Q/Q_f for shape 1	0
QNORM(I,2)	Second value of Q/Q_f for shape 1	0
■	■	
QNORM(I,MM(I))	Last value of Q/Q_f for shape 1	0
	(Total of MM(17)/8 + MM(18)/8 groups)	
	REPEAT GROUP E1 FOR EACH NUMBERED SEWER ELEMENT (maximum number of elements = 200). THESE GROUPS MAY BE READ IN ANY ORDER.	
	Sewer Element Data	
	Variables with asterisks can be modified using the Default/Ratio option.[13,14]	
E1	Group identifier	None
NOE	External element number.[12] No element may be labeled with a number greater than 10,000, and it must be a positive numeral. However, numbering need not be consecutive or continuous.	None
	EXTERNAL[12] NUMBER(S) OF UPSTREAM ELEMENT(S). UP TO THREE ARE ALLOWED. A ZERO DENOTES NO UPSTREAM ELEMENT (maximum value = 10,000)	
NUE(1)	First of three possible upstream elements.	0
NUE(2)	Second of three possible upstream elements.	0
NUE(3)	Third of three possible upstream elements.	0
NTYPE	Classification of element type. Obtain value from Table 6-2 or 6-4.	0

Table 6-7. Continued

Variable	Description	Default
	THE FOLLOWING VARIABLES ARE DEFINED BELOW FOR CONDUITS ONLY. REFER TO TABLE 6-4 FOR REQUIRED INPUT FOR NON-CONDUITS.	

Requirements for natural channels (type 16) are indicated in **bold face**. Only required parameters are SLOPE and BARREL (ID number). All other data are entered in data groups E2 - E4.

Variable	Description	Default
DIST*	Element length for conduit, ft [m]. (For manhole, constant inflow into system, cfs [m^3/sec].) **Not required (N.R.) for type 16.**	0.0
GEOM1*	First characteristic dimension of conduit, ft [m]. See Figure 6-3 and Table 6-3 for definition. (For manhole: constant concentration of pollutant 1 in the inflow if simulated. Units according to NDIM, group F1). **N.R. for type 16.**	0.0
SLOPE*	Invert slope of conduit, ft/100 ft (i.e., percent). (For manhole: constant concentration of pollutant 2 in the inflow, if simulated. Units according to NDIM, Group F1.) **Required for type 16.**	0.0
ROUGH*	Manning's roughness of conduit. (For manhole: constant concentration of pollutant 3 in the inflow, if simulated. Units according to NDIM, Group F1.) **N.R. for type 16.**	0.0
GEOM2*	Second characteristic dimension of conduit, ft [m]. See Figure 6-3 and Table 6-3 for definition. (N.R. for some conduit shapes.) (For manhole: constant concentration of pollutant 4 in the inflow, if simulated. Units according to NDIM, group F1).	0.0
BARREL	Number of barrels for this element. The barrels are assumed to be identical in shape and flow characteristics.[15] (Must be integer $\geq$ 1.) **For type 16, the cross-section ID number (SECNO, group E3) of the cross section for this channel.**	1.0
GEOM3*	Third characteristic dimension of conduit, ft [m]. See Figure 6-3 and Table 6-3 for definition. (N.R. for some conduit shapes.)	0.0

The E2 (NC), E3 (X1), and E4 (GR) data lines for any type 16 elements follow as a group after all E1 lines have been entered.

Data groups E2, E3 and E4 correspond to HEC-2 lines NC, X1 and GR. HEC-2 input may be used directly if desired. Lines may be identified either by Transport Block identifiers (E2, E3, E4) or HEC-2 identifiers (NC, X1, GR).

Channel Roughness

This is an optional data line that permanently modifies the Manning's roughness coefficients (n) for the remaining natural channels. This data group may repeated for later channels. It <u>must</u> be included for the first natural channel modeled. XNCH is used to normalize the cross-sectional flow-area relationship.

Variable	Description	Default
E2 or NC	Group identifier	None
XNL	n for the left overbank. = 0.0, No change, > 0.0, New Manning's n.	0.0
XNR	n for the right overbank. = 0.0, No change, > 0.0, New Manning's n.	0.0
XNCH	n for the channel. = 0.0, No change, > 0.0, New Manning's n.	0.0

Table 6-7. Continued

Variable	Description	Default
	Cross Section Data	
	Required for type 16 elements in earlier E1 data lines. Enter pairs of E3 and E4 lines.	
E3 or X1	Group identifier	None
SECNO	Cross section identification number.	1
NUMST	Total number of stations on the following E4 (GR) data group lines. NUMST must be < 99.	0
STCHL	The station of the left bank of the channel, ft [m]. Must be equal to one of the STA(N) on the E4 (GR) data lines.	0.0
STCHR	The station of the right bank of the channel, ft [m]. Must be equal to one of the STA(N) on the E4 (GR) data lines.	0.0
XLOBL	Not required for Transport (enter 0.0).	0.0
XLOBR	Not required for Transport (enter 0.0).	0.0
LEN(N)	Length of channel reach for this cross section, ft [m].	0.0
PXSECR	Factor to modify the horizontal dimensions for a cross section. The distances between adjacent E4 (GR) stations (STA) are multiplied by this factor to expand or narrow a cross section. The STA of the first E4 (GR) point remains the same. The factor can apply to a repeated cross section or a current one. A factor of 1.1 will increase the horizontal distance between the E4 (GR) stations by 10 percent. Enter 0.0 for no modification.	0.0
PSXECE	Constant to be added (+ or -) to E4 (GR) elevation data on next E4 (GR) line. Enter 0.0 to use E4 (GR) values as entered.	0.0
	Cross-Section Profile	
	Required for type 16 elements in data group E1. Enter E3 and E4 lines in pairs.	
E4 or GR	Group identifier	None
EL(1)	Elevation of cross section at STA(1). May be positive or negative, ft [m].	0.0
STA(1)	Station of cross section 1, ft [m].	0.0
EL(2)	Elevation of cross section at STA(2), ft [m].	0.0
STA(2)	Station of cross section 2, ft [m].	0.0

Enter NUMST elevations and stations to describe the cross section. Enter 5 pairs of elevations and stations per data line.[16] Stations should be in increasing order progressing from left to right across the section. Cross section data are traditionally oriented looking downstream (HEC, 1982).

	SKIP TO GROUP G1 IF NPOLL = 0 (GROUP B1)	
	Quality Input Data	
F1	Group identifier	None
KPOL	Pollutant selector from interface file e.g., if KPOL = 7, seventh constituent on interface file will be this pollutant. User must know contents on interface file from preceding block. If KPOL = 0, this pollutant is defined below and not taken from interface file.[17]	0
	The following three parameters, PNAME, PUNIT, NDIM are required if KPOL is equal to 0. If KPOL is > 0 enter nothing for PNAME, PUNIT and NDIM and skip to parameter DECAY.	
PNAME	Pollutant name[18] (Character data - 8 columns).	None
PUNIT	Pollutant units[18] (Character data - 8 columns).	None

Table 6-7. Continued

Variable	Description	Default
NDIM	Type of units[18]	0
	= 0, mg/l.	
	= 1, "other" per liter, e.g., MPN/l.	
	= 2, other concentration units, e.g. pH, JTU.	
DECAY	First order decay coefficient, day^{-1}	0.0
SPG	Specific gravity. If SPG > 1.0, pollutant will be subject to scour-deposition calculations.	0.0
	The following parameters are <u>not entered</u> if SPG <= 1.0.	
	Particle Size Distribution. First point, PSIZE(1) = 0.0 mm, PGR(1) = 100.0 is automatically included. See Figure 6-5.[19]	
PSIZE(2)	Particle size, mm.	0.0
PGR(2)	Percent greater than, %	0.0
PSIZE(3)	Particle size, mm	0.0
PGR(3)	Percent greater than, %	0.0
PSIZE(4)	Particle size, mm	0.0
PGR(4)	Percent greater than, %	0.0
PSIZE(5)	Particle size, mm	0.0
PGR(5)	Percent greater than, %	0.0
PSDWF	Maximum particle size contained in dry-weather flow input (either through manholes or using subroutine FILTH). MUST BE <= PSIZE(5).	0.0
	GROUPS G1 THROUGH G5 ARE FOR INTERNAL STORAGE (NTYPE = 22). OMIT IF INTERNAL STORAGE IS NOT DESIRED AND SKIP TO GROUP H1. REPEAT GROUPS G1-G5 FOR EACH INTERNAL STORAGE ELEMENT, IS.	
	(Maximum of 30 storage elements.)	
G1	Group identifier	None
LOUT(IS)	Outflow routing parameter	0
	= 0, the depth-outflow relationship is described by as many as 16 data pairs on the G2 cards.	
	= 1, the depth-outflow relationship is described by a single power equation in Group G3.	
	= 2, the depth-outflow relationship is governed by two power equations in Group G3.	
	= 3, the depth-outflow relationship is controlled by the pumps described in Group G4.	

<div align="center">Depth - Surface Area - Volume - Outflow Data</div>

Each line contains a column of a unit depth and the corresponding values of area, volume and treated (e.g., weir overflow) outflow. The column for outflow may be left blank depending on the value of LOUT(IS) in Group G1. If no values for volume are entered, the program estimates volume from the depth-surface area relationship. Order the lines from the bottom of the unit (TSDEP(IS,1) = 0.0) to the maximum depth including as many as sixteen lines.

G2	Group identifier	None
TSDEP(IS,MM)	A unit depth, ft [m].	0.0
TSAREA(IS,MM)	Surface area corresponding to the above depth, ft^2 [m^2]	0.0
TSTORE(IS,MM)	Volume corresponding to the above depth, ft^3 [m^3]	0.0
TSQOU(IS,MM)	Outflow at the above depth, cfs [m^3/sec]	0.0

Table 6-7. Continued

Variable	Description	Default
	Depth-Outflow Power Equation	
	Required only if LOUT(IS) = 1 or 2 (Group G1).	
	See Equation 7-3 for definition of coefficient, depth, and exponent.	
G3	Group identifier	None
A1(IS,1)	Depth-outflow equation coefficient. (Units depend on A2 and whether ft or m are used.)	0.0
DO(IS,1)	Depth-outflow equation minimum flow depth, ft [m].	0.0
A2(IS,1)	Depth-outflow equation exponent.	0.0
	The following parameters required only if LOUT(IS) = 2 (Group G1).	
A1(IS,2)	Depth-outflow equation coefficient for second outlet. (Units depend on A2 and whether ft or m are used.)	0.0
DO(IS,2)	Depth-outflow equation minimum flow depth for second outlet, ft [m].	0.0
A2(IS,2)	Depth-outflow equation exponent for second outlet.	0.0
GEOM3	External element number into which flows the outflow from the second outlet.	0.0
	Outflow Pumping	
	Required only if LOUT(IS) = 3 (Group G1)	
G4	Group identifier	None
TDSTAR(IS,1)	Depth at which pumping rate TQPUMP(IS,1) begins, ft. [m].	0.0
TDSTAR(IS,2)	Depth at which pumping rate TQPUMP(IS,2) begins, ft. [m]. Must be $\geq$ TDSTAR(IS,1)	0.0
TQPUMP(IS,1)	Pumping rate when depth $\geq$ ft [m]. TDSTAR(IS,2) cfs [m^3/sec].	0.0
TDSTOP(IS)	Depth below which all pumping stops ft [m]. Must be $\leq$ TDSTAR(IS,1)	0
	Initial conditions in internal storage element IS	
G5	Group identifier	None
STORL(IS)	Total volume of water in unit at the start of the simulation, ft^3 [m^3]	0.0
	The initial pollutant concentrations are required only if STORL(IS) > 0.0. The concentrations must be given with dimensions consistent with those entered in Group F1.	
PTCO(IS,1)	Concentration of pollutant in the storage unit at the start of the simulation. Required only if NPOLL $\geq$ 1.	0.0
PTCO(IS,2)	Concentration of pollutant in the storage unit at the start of the simulation. Required only if NPOLL $\geq$ 2.	0.0
PTCO(IS,3)	Concentration of pollutant 3 in the storage unit at the start of the simulation. Required only if NPOLL $\geq$ 3.	0.0
PTCO(IS,4)	Concentration of pollutant 4 in the storage unit at the start of the simulation. Required only if NPOLL = 4.	0.0
	SKIP TO GROUP I1 IF NOUTS = 0 IN GROUP B1	
	List of external non-conduit element numbers for which outflows are to be transferred to subsequent blocks for a total of NOUTS (Group B1) non-conduit elements.[20]	
H1	Group identifier	None
JN(1)	First element number[21]	0
■	■	
JN(NOUTS)	Last element number	0

Table 6-7. Continued

Variable	Description	Default
	SKIP TO GROUP J1 IF NINPUT =0 (GROUP B1)	
	Non-conduit element numbers into which hydrographs and pollutographs (from input using group R1) enter the sewer system. These must be the order in which hydrograph and pollutograph ordinates appear at each time step in group R1. A total of NINPUT values required.[20]	
I1	Group identifier	None
NORDER(1)	First element number	0
■	■	
NORDER(NINPUT)	Last element number	0
	SKIP TO GROUP J2 IF NNYN = 0 (GROUP B1)	
	List of external non-conduit element numbers at which input hydrographs and pollutographs are to be stored and printed out, for a total of NNYN (Group B1) non-conduit elements.[20]	
J1	Group identifier	None
NYN(1)	First input location number	0
■	■	
NYN(NNYN)	Last input location number	0
	SKIP TO GROUP K1 IF NNPE = 0 (GROUP B1)	
	List of external non-conduit element numbers at which output hydrographs and pollutographs are to be stored and printed out for a total of NNPE (Group B1) non-conduit elements.[20]	
J2	Group identifier	None
NPE(1)	First output location number	0
■	■	
NPE(NNPE)	Last output location number	0
	IF SUBROUTINE INFIL IS TO BE CALLED (NINFIL = 1), INSERT GROUPS K1 THROUGH K2, OTHERWISE OMIT.	
	Estimated Infiltration	
K1	Group identifier	None
DINFIL	Base dry weather infiltration, cfs [m^3/sec].	0.0
GINFIL	Groundwater infiltration, cfs [m^3/sec].	0.0
RINFIL	Rainwater infiltration, cfs [m^3/sec].	0.0
RSMAX	Peak residual moisture, cfs [m^3/sec].	0.0
	Constant concentrations of pollutants in infiltration. Not required if NPOLL = 0. Units of each according to NDIM, Group F1.	
CPINF(1)	Concentration of pollutant 1	0.0
CPINF(2)	Concentration of pollutant 2	0.0
CPINF(3)	Concentration of pollutant 3	0.0
CPINF(4)	Concentration of pollutant 4	0.0

Table 6-7. Continued

Variable	Description	Default
	Monthly Degree-Days[22]	
K2	Group identifier	None
NDD(1)	January degree-days, $\diamond$F-day, [no metric units]	0.0
NDD(2)	February degree-days	0.0
■	■	
NDD(12)	December degree-days	0.0
	IF SUBROUTINE FILTH IS TO BE CALLED (NFILTH = 1), INSERT GROUPS L1 THROUGH Q1, OTHERWISE OMIT	
	Factors to correct yearly average sewage flows to daily average by accounting for daily variations throughout a typical week.	
L1	Group identifier	None
DVDWF(1)	Flow correction for Sunday	1.0
DVDWF(2)	Flow correction for Monday	1.0
■	■	
DVDWF(7)	Flow correction for Saturday	1.0
	IF NPOLL = 0, SKIP TO GROUP M1. (NOTE: IF POLLUTANTS ARE SIMULATED AND FILTH IS CALLED, FIRST THREE POLLUTANTS MUST BE BOD5, SUSPENDED SOLIDS AND TOTAL COLIFORMS[23].	
	Factors to correct BOD yearly averages to daily averages.	
L2	Group identifier	None
DVBOD(1)	BOD correction for Sunday	1.0
■	■	
DVBOD(7)	BOD correction for Saturday	1.0
	Factors for correction of yearly SS averages to daily averages	
L3	Group identifier	None
DVSS(1)	SS correction for Sunday	1.0
■	■	
DVSS(7)	SS correction for Saturday	1.0
	(No daily correction factors for coliforms)	
	Factors to correct daily average sewage flow to hourly averages by accounting for hourly variations throughout a typical day. Enter all factors on one line.	
M1	Group identifier	None
HVDWF(1)	Midnight to 1 a.m.	1.0
■	■	
HVDWF(24)	11 p.m. to midnight factor.	1.0
	IF NPOLL = 0, SKIP TO GROUP N1	
	Factors for BOD hourly corrections (one line).	
M2	Group identifier	None
HVBOD(1)	Midnight to 1 a.m. factor.	1.0
■	■	
HVBOD(24)	11 p.m. to midnight factor.	1.0

Table 6-7. Continued

Variable	Description	Default
	Factors for SS hourly corrections (1 line).	
M3	Group identifier	None
HVSS(1)	Midnight to 1 a.m. factor.	1.0
■	■	1.0
HVSS(24)	11 p.m. to midnight factor.	
	Factors for total coliform hourly corrections (1 line).	
M4	Group identifier	None
HVCOLI(1)	Midnight to 1 a.m.	1.0
■	■	
HVCOLI(24)	11 p.m. to Midnight factor.	1.0
	Study Area Data	
N1	Group identifier	None
KTNUM	Total number of subareas within a given study area in which sewage flow and quality are to be estimated.	1
KASE	Indicator as to whether study area data, such as treatment plant records, are to be used to estimate sewage quality? = 1, Yes = 2, No	KASE
NPF	Total number of process flows within the study area for which data are included in one of the following groups.	0
KDAY	Number indicating the day of the week during which simulation begins (Sunday = 1).	1
CPI	Consumer Price Index	125.0
CCCI	Composite Construction Cost Index	110.0
POPULA	Total population in all areas, thousands.	None
	IF KASE = 1, INCLUDE GROUPS O1,O2, AND P1	
	Average study area data[24]	
O1	Group identifier	None
ADWF	Total study area average sewage flow e.g., from treatment plant records, cfs [m^3/sec].	0.0
ABOD	Total study area average BOD,mg/l	0.0
ASUSO	Total study area average SS, mg/l	0.0
ACOLI	Total study area average coliforms MPN/100 ml.	0.0
	Categorized study area data.	
O2	Group identifier	None
TOTA	Total study area from which ABOD and ASUSO were taken, acres [ha].	0.0
TINA	Total contributing industrial area, acres [ha].	0.0
TCA	Total contributing commercial area, acres [ha].	0.0
	*** Valuations for the following three parameters are for 1963 dollars! ***	
TRHA	Total contributing high income (above \$15 000) residential area, acres [ha]	0.0
TRAA	Total contributing average income (above \$7000 but below \$15 000) residential area, acres [ha].	0.0

Table 6-7. Continued

Variable	Description	Default
TRLA	Total contributing low income (below $4 000) residential area, acres [ha].	0.0
TRGGA	Total area from the above three residential areas that contribute additional waste from garbage grinders, acres [ha].	0.0
TPOA	Total park and open area within the study area, acres [ha].	0.0
	IF PROCESS FLOW DATA ARE AVAILABLE (NPF NOT EQUAL 0 AND KASE = 1), REPEAT GROUP P1 FOR EACH PROCESS FLOW (NPF DATA). OTHERWISE, SKIP TO GROUP Q1.	
	Process flow characteristics	
P1	Group identifier	None
INPUT	External manhole number into which flow is assumed to enter (maximum value = 10,000, minimum value = 1).	0
QPF	Average daily process flow entering the study area system, cfs [m^3/sec].[26]	0.0
BODPF	Average daily BOD of process flow, mg/l.	0.0
SUSPF	Average daily SS of process flow, mg/l.	0.0
	REPEAT GROUP Q1 FOR EACH OF THE KNUM SUBAREAS. THESE SUBAREAS DO NOT NECESSARILY HAVE TO CORRESPOND TO RUNOFF SUBCATCHMENTS.	
	Subarea data	
Q1	Group identifier	None
KNUM	Subarea number.	0
INPUT	External number of the manhole into which flow is assumed to enter for subareas KNUM (Maximum value = 10,000, minimum value = 1).	0
KLAND	Predominant land use within subarea = 1, single family residential = 2, multi-family residential = 3, commercial = 4, industrial = 5, undeveloped or park lands	5
METHOD	Parameter indicating whether or not water usage within subarea KNUM is metered. = 1, metered water use = 2, incomplete or no metering	2
KUNIT	Parameter indicating units in which water usage estimates (WATER) are tabulated = 0, thousand gal/mo. [10^3 cu.m/mo] = 1, thousand cu.ft./mo [10^3 cu.m/mo]	0
MSUBT	Subtotal printed after each subarea? = 0, No = 1, Yes	0
SAGPF	Total industrial process flow originating within subarea KNUM, cfs [m^3/sec][25]	0.0
SABPF	BOD contributed from industrial process flow originating within subarea KNUM, mg/l	0.0
SASPF	SS contributed from industrial process flow originating within subarea KNUM, mg/l	0.0

Table 6-7. Continued

Variable	Description	Default
	Several of the following parameters are optional. See text.	
WATER	Measured winter water use for subarea KNUM in the units specified by KUNIT.[26]	0.0
PRICE	Cost of the last thousand gal [10^3 cu.m] of water per billing period for an average consumer within subarea KNUM, cents/1000 gal [cents/10^3 cu.m]	0.0
SEWAGE	Measured average sewage flow from entire subarea KNUM, cfs [cu.m/sec] but not including process flows (SAQPF).[26]	0.0
ASUB	Total area within subarea KNUM, acres [ha].	0.0
	The next six parameters are not required if KLAND > 2.	
POPDEN	Population density within subarea KNUM, persons/acre [pers/ha].[27]	0.0
DWLNGS	Total number of dwelling units within subarea KNUM.[28]	10.0/ac
FAMILY	Number of people living in average dwelling unit within subarea KNUM.[28]	3.0
VALUE	Market value of average dwelling unit within subarea KNUM, thousands of dollars.	20.0
PCGG	Percentage of dwelling units possessing garbage grinders within subarea KNUM	0.0
XINCOM	Income of average family living within subarea KNUM, thousands of dollars per year.	VALUE/2.5
	END OF FILTH DATA CARDS	
	Include group R1 only if NINPUT not = 0 (Group B1)	
	REPEAT GROUP R1 FOR EACH INLET FOR FIRST TIME AND THEN REPEAT GROUP R1 FOR EACH INLET FOR SECOND TIME, ETC. REPEAT THIS COMBINATION FOR ALL INPUT TIMES. ORDER OF INLET GROUP MUST BE THE SAME AS INDICATED IN GROUP I1.	
	Hydrograph and pollutograph input ordinates	
R1	Group identifier	None
TEO	Time of day, decimal hours, e.g., 6:30 p.m. = 18.5. The first time must equal TZERO. The program will automatically set TEO = TZERO for entries for the first time.[29]	0.0 or TZERO
QE2	Input flow for this time step at first inlet, cfs [m^3/sec].	0.0
	*** Enter pollutant concentration only for NPOLL pollutants *** (Enter no fields if NPOLL = 0)	
PE2(1)	Pollutant 1 for this time at first inlet, concentration according to NDIM (Group F1).	0.0
PE2(2)	Pollutant 2 for this time at first inlet, concentration according to NDIM (Group F1).	0.0
PE2(3)	Pollutant 3 for this time at first inlet, concentration according to NDIM (Group F1).	0.0
PE2(4)	Pollutant 4 for this time at first inlet, concentration according to NDIM (Group F1).	0.0
	END OF TRANSPORT BLOCK DATA GROUPS	
	At this point the program seeks new input from the Executive Block.	

Footnotes to Table 6-7

1. The title from the first of any preceding blocks run will also be printed.

2. Except in unusual cases, these errors will only indicate that a small continuity violation will occur. These errors can usually be cured by shortening the time step or increasing the length of the conduit.

3. The Transport Block time step does not have to equal that of a preceding block. The total simulation time is NDT$\diamond$DT. If this is greater than that of an input file, the simulation will end earlier. Although there is no limit on the length of a simulation, output is geared toward single events. That is, daily or monthly totals are not printed and zeroes are not suppressed.

4. If the time step for Transport is different than that for a preceding block, input hydrograph and pollutograph ordinates will be found by linear interpolation at the required time.

5. If both local (keyboard) and interface file input is used, hydrographs and pollutograph loads entered at common locations will be summed. Transport can be run without hydrograph input if it were desired to route only dry-weather flows (base flows).

6. Constant base flows and concentrations may be entered at manhole elements (Type 19) in group E1 if desired, thus eliminating the need to call FILTH or INFIL. They may still be called if desired; infiltration, dry-weather flow and base flow will be summed for entry at non-conduits. Base flows and concentrations entered in group E1 will not be subject to daily and hourly correction factors.

7. Inlet hydrographs from a preceding block will automatically be accepted, but a match must be found for each inlet (element) number on the interface file with an element number entered in Transport.

8. These locations should include any elements for which graphical output is desired since only locations on the interface file may be plotted using the Executive Block.

9. A/A_f = ANORM is the cross-sectional flow area divided by the cross-sectional flow area of the pipe running full. Tabular values of ANORM are generated in the program by dividing the ANORM axis (0.0,1.0) into NN1 or MM1 equal divisions.

10. y/y_f = DNORM is the depth of flow, y, divided by the maximum flow depth, y_f (e.g. diameter of a circular conduit).

11. Q/Q_f = QNORM is the flow rate divided by the flow rate of the conduit flowing full.

12. "External" numbers are those assigned by the user to the various sewer system components. "Internal" numbers are subscripts assigned within the program in the order in which elements in Group E1 are entered. All input to the Transport Block is in terms of external numbers.

13. Input values in this group indicated with asterisks are multiplied by ratios, initially set equal to 1.0. If the ID number (NOE) = -1, non-zero entries for parameters with asterisks will replace old values of the ratios. These will become new ratios for all future entries of these parameters. Ratios may be altered or reset to 1.0 any number of times. The intention of the use of ratios is to simplify sensitivity analyses, etc., by allowing easy changes of data values without editing data. The altered ratios apply to all subsequent input of a given parameter (until changed by another default/ratio line).

14. Input parameters in this group indicated with asterisks will take on default values if input values are zero. If the ID number (NOE) = -2, non-zero data entries for parameters with asterisks will become new default values for all future entries of these parameters. Default values may be altered or reset to their original values (except zero) any number of times. It is not possible to reset a default value exactly to zero since only non-zero values are changed. However, the value may be made arbitrarily small, for example, 1.0E-50. The altered default values apply to all subsequent input of a given parameter (until changed by another default/ratio line).

15. For example, a two-barreled conduit would consist of two identical parallel conduits adjacent to each other, as in a double box culvert.

16. Enter the group identifier (E4) on every line of input.

17. If NFILTH = 1 (Group D3), it will be assumed that the first pollutant is BOD5, the second is suspended solids and the third is total coliforms. Hence, the selection indicated by NPOLL must be in this order. The fourth pollutant, if simulated, is unaffected by the value of NFILTH.

18. See the discussion for the Runoff Block.

19. Up to the five available points (including 0.0 mm, 100%) may be used to define the particle size distribution. For instance, if a triangular distribution were satisfactory with 0.1 mm being the largest particle size, then PSIZE(2) = 0.1, PGR(2) = 0.0 would be the only entries needed. (But trailing zeroes are still needed to complete the free-format input.)

20. May require multiple lines of input.

21. Be careful in subsequent blocks to ensure that element numbers will correspond to those transferred in the interface file. However, any element number may be placed on the interface file, e.g., for plotting purposes.

22. See Table VIII-1, Appendix VIII, for representative values. Degree days are not needed if RSMAX = 0.0 in Group K1 (but 12 zeroes must still be included for Group K2). Degree days are in °F-day, and there is no metric input for this parameter.

23. When subroutine FILTH is used to generate dry-weather flow and pollutants, the only pollutants included are BOD5, suspended solids (SS) and total coliforms (T.C.). NPOLL must be 3 or 4, and the three pollutants entered in the given order in group F1. The fourth pollutant is arbitrary (if used) and is not affected by FILTH calculations.

24. Predicted total average daily dry-weather flow from downstream end(s) of the system will be adjusted to these values.

25. These industrial process flows described by SAQPF, SABPF and SASPF will be added to flows and quality already generated.

26. If process flows are too large it is possible for ADWF - infiltration - QPF < 0 in which case the program defaults to KASE = 2.

27. Either SEWAGE or else WATER is required to generate dry-weather flow from commercial areas. If both are zero, only process flows will be considered. SEWAGE and WATER should = 0, for industrial land use; for KLAND = 4, let entire flow be process flow.

28. DWLNGS and FAMILY can be used to calculate the population and the population density. If both are given, the value of POPDEN will be overridden.

29. The program will interpolate linearly between entries of TE2 to obtain intermediate values of flow and concentrations. Hence, the difference between two time entries, TE2, should not be less than the time step, DT, unless a step function change is desired. Time entries need not be equally spaced, but the last time entered must extend past the end time of the run. Time TZERO is read from the interface file or else entered in Group D2. If simulation extends beyond midnight (i.e., TE2 > 24), continue the running time into the next day or days (i.e., let TE2 be greater than 24). Note that the time TE2 must be the same for all inlets. (The program will use the value for TE2 entered on the line for the last inlet.)

Section 7
Storage/Treatment Block[*]

Block Description
Introduction

The Storage/Treatment Block has been developed to simulate the routing of flows and pollutants through a dry- or wet-weather storage/treatment plant containing up to five units or processes. The block will accept any number of time steps; therefore, a single-event or a continuous simulation is possible. Each unit may be modeled as having detention or non-detention characteristics. The various units may be linked in a variety of configurations. Sludge handling may also be modeled using one or more units. Additionally, capital cost and operation and maintenance cost may be estimated for each unit.

The S/T Block will route, in addition to flow, up to three different pollutants. These pollutants may be input to the block from any external block via the interface file, directly from keyboard input to this block, or a combination of both. Characterization of the pollutants may be by magnitude (i.e., concentration) or by magnitude and a particle size/specific gravity settling velocity distribution. All input flows and pollutant concentrations are assumed to be instantaneous values. However, the instantaneous values at the beginning and end of each time are used to compute average values for each time step. Thus, the user is cautioned that the output from the S/T Block consists of average values, not instantaneous values as in the rest of SWMM. This section describes the program operations of the S/T Block, provides instructions for preparing input data groups, defines program variables, and presents test applications. Theoretical development and explanations are given in Appendix IV.

Program Operation

The Storage/Treatment Block consists of approximately 2000 Fortran statements grouped in eight subroutines. The relationships among the S/T Block, the rest of SWMM and the various subroutines are shown in Figure 7-1.

Subroutine STRT is called by the Executive Block to initiate the operation of the S/T Block. STRT provides the main driving loop for the block and generally acts as the central coordinating subroutine. Input flow and pollutants from any external block are read in this subroutine. Output from the S/T Block transferred to other blocks is also handled by STRT (through the Executive Block). The information transferred to other blocks is discussed in the next paragraph. Subroutine STRDAT is called in STRT at the start of a run and is responsible for reading the input data describing the units, their configuration, the pollutant removal mechanisms,

---

[*] This block was originally developed by Dr. Stephen J. Nix.
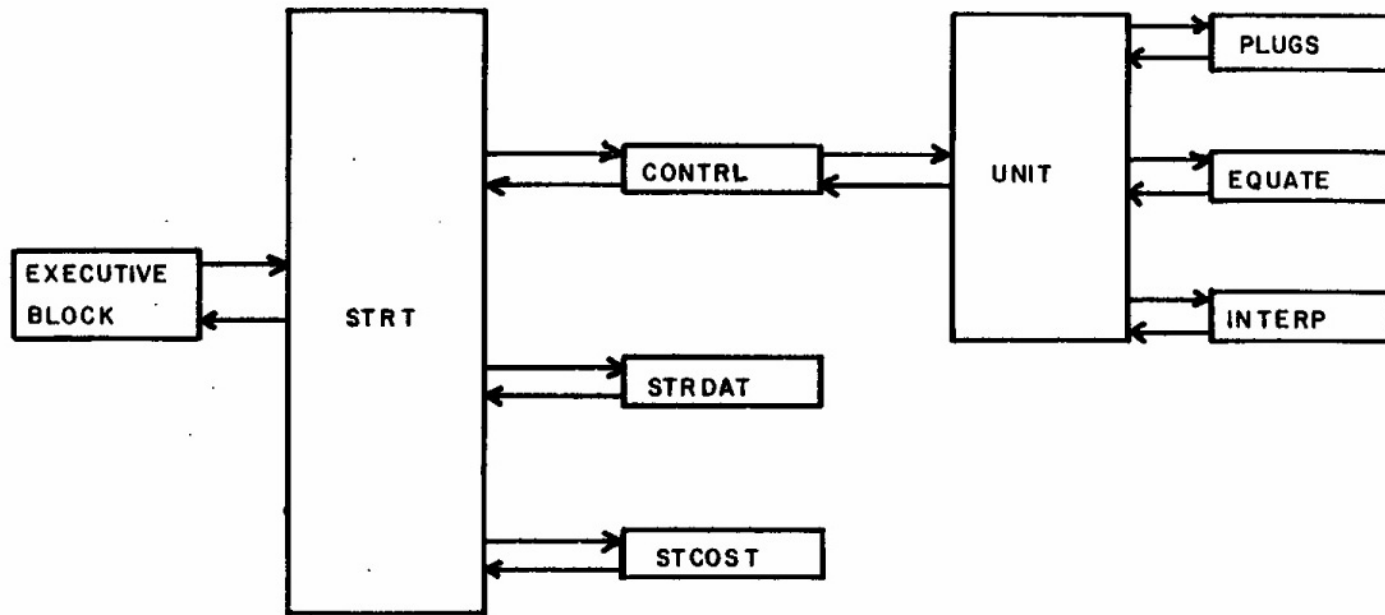
Figure 7-1. Storage/Treatment Block.

the method of characterizing pollutants and the remainder of the data provided by the user (excluding flow and pollutant inputs). STRDAT also prints the input data for verification. Subroutine CONTRL is called each time step from the main driving loop in Subroutine STRT. CONTRL directs flow and pollutants between units and any subsequent block. CONTRL also coordinates the accounting and printout of detailed and summarized performance information. Flow and pollutant inputs from input data groups (or data "lines") are also read in this subroutine.
Subroutine UNIT is called from Subroutine CONTRL for each unit and is the heart of the S/T Block. UNIT has the flexibility and capability to model detention and non-detention units with a variety of pollutant removal mechanisms, residual removal schemes and outflow structures. Subroutine EQUATE is used by UNIT to provide a variety of pollutant removal equations. Subroutine INTERP is employed by UNIT for linear interpolation in routing flows through detention units. Subroutine PLUGS is used by UNIT to model pollutant routing through a detention unit when perfect plug flow is specified. Subroutine STCOST is called from STRT to estimate the capital cost and operation and maintenance cost for each unit.

Use of Off-Line Computer Storage

No scratch data sets are required to run the Storage/Treatment Block. However, disks files may be used as an input source (from an external block) or to transfer output to other blocks. This interface file consists of descriptive titles, user-supplied pollutant names and dimensions, the simulation starting data and time, the name of the external block generating the output (input) file, the total catchment or tributary area, the number of elements (inlets, outfalls, non-conduits, etc.) and pollutants found on the output (input) file, and the elements for which flow and pollutant data are placed (read from) the output (input) file. This preliminary information is followed by the flows and pollutant data for each time step (up to the total number of time steps) for each of the specified elements. The user should refer to Section 2 for a detailed description of the interface file.

Flow and up to three pollutants are transferred unchanged from an input file to an output file if the data are from locations or are pollutants not selected for use in the S/T Block. In other words, the flow and pollutant data from an input file (external block) are altered only if they pass through a storage/treatment facility. However, the dimensions of the data on the input file will be made to conform to the standard SWMM dimensions before being placed on an output file (see Section 2). The methods of selecting these data from particular location (i.e., inlet, element) numbers and processing specific pollutants is discussed in the next subsection.

Importance of Data

The Storage/Treatment Block is a user-intensive model, i.e., the user should have a thorough knowledge of what he/she is modeling. This is an obvious, but often overlooked, axiom and especially true when a model provides some default values. The user is required to describe each unit in some detail with data of his/her own choosing. S/T performance depends on local pollutant characteristics, flow rates, etc.; thus, the user is encouraged to use local operating data, whenever possible, to aid in the modeling effort.

Instructions for preparing the input data are presented below with suggestions and examples. All entries not described in the text are considered to be self-explanatory or covered sufficiently in Table 7-4, which gives the input order. All data groups are required unless otherwise stated. The general structure of the data groups is shown in Table 7-1.

Table 7-1. Structure of Data Groups for the Storage/Treatment Block

Data Group	Type of Data
A1	Title
B1	General data
C1 - C2	Starting time and print instructions
D1	Evaporation
E1 - E6	Pollutant characterization
F1 - F2	General unit characteristics
G1 - G4	Pollutant removal
H1 - H8	Detention units
I1	Costs
J1	Flow and pollutants

Preliminary Information

Title (Data Group A1)

This data group allows a descriptive heading to be displayed at the start of the printed output. The heading is also transferred to the next block. Two lines are required.

General Information (Data Group B1)

The variable NOTAPE allows the user to specify the source of flow and pollutants used for input to the S/T Block. If NOTAPE = 0, the input is provided by an external input file (arranged by the Executive Block). When NOTAPE = 1, the input is provided by data group J1. If NOTAPE = 2, the input is the sum of the entries for each time step from the external file and data group J1. The parameter JNS selects the location number (from an external file, NOTAPE = 0 or 2) from which flows and pollutants will be taken and passed through the S/T Block. If the input is provided solely by data group J1 (NOTAPE = 1 or 2), the user may label the output of the S/T Block with a location number.

When NOTAPE = 0 or 2, the variables NDT (number of time steps) and DS (time step) are read from the external input file. However, the value of NDT from an external block may be altered by specifying a non-zero value on data line B1. There is no limit on the number of time steps. This is useful for extending the simulation beyond the limit of the external block or input file. The value of DS from an external block will supersede any value for DS entered on data line B1. If run independently (NOTAPE = 1), there is no restriction on time step size for the S/T Block. However, if DS is too large, Puls routing may result in too-rapid drawdown of detention units.

The variable NU specifies the number of storage/treatment units, including the residual (sludge) handling units, to be modeled and the number of times that data groups F1 through I1 are repeated. The model as written is limited to <u>five units.</u> The variable NP specifies the number of pollutants to be routed through the storage/treatment system. If NP = 0, then only flows are routed. A maximum of three pollutants may be selected for routing. The specific pollutants are

selected on data line E1. However, NP may not be greater than the number of pollutants transferred from other blocks (NOTAPE = 0 or 2).

If ICOST = 1, the capital and operation and maintenance costs are computed for each unit. Cost equation parameters are entered in data group J1. The value of METRIC determines whether U.S. customary or metric units are used for all data input, output and internal calculations. The value of TRIBA is the service area of the S/T plant. It does not enter into the computations but is transferred to the next block.

Starting Time and Print Instructions (Groups C1 and C2)

The values of IDATE and ITIME are used to start the date/time algorithm of the S/T Block. The Julian date and time of day in seconds are updated for each time step. If NOTAPE = 0 or 2 (data group B1) the values of IDATE and ITIME read from the external file supersede the values entered on data group C1.

The user is cautioned against printing large quantities of unwanted information. The first runs should have as little printout as possible (ISUM = 0) and IDET = 0) to check for obvious errors in the input data. As simulation efforts proceed, more detailed printouts may be desired.

Data group C2 (one line) is used to enter the first and last dates of the detailed print periods specified by NPR (data line C1).

Evaporation Data (Group D1)

Monthly evaporation rates are required to correct for evaporation from detention units. However, two groups must be included even if there are no detention units. If needed, values of evaporation (parameters E(1) through E(12)) are entered for the months during which the simulation occurs; others may be left blank.

Pollutant Characterization (Groups E1 - E6)

Data groups E1 through E6 are omitted if NP = 0 (data group B1).

Data Group E1

These groups (up to two) allow the user to select the NP (data group B1) pollutants to be routed through the S/T Block. The variables IPOLL(1), IPOLL(2), and IPOLL(3) are used to select pollutants from an external input file (NOTAPE = 0 or 2, data group B1). For example, if IPOLL(1) = 5, then the first pollutant routed through the S/T Block is the fifth pollutant found on the input file. If data from data group J1 are to be added to the external input file data (NOTAPE = 2, data group B1), the pollutants entered on that data group must be in the same order as specified by IPOLL(1), IPOLL(2), and IPOLL(3).

Pollutants may be characterized by their magnitude alone (concentration) or by their magnitude and a particle size/specific gravity or settling velocity distribution by specifying IPART(IP) = 0 or 1, respectively. If IPART(IP) = 1, the user is required to enter these distributions on data groups E2 through E6. Using these distributions, however, limits the type of
unit that may be modeled (discussed in later data groups.)

The variables NDIM(1), NDIM(2), and NDIM(3) are used to describe the dimensions of the pollutant data on data group J1 when NOTAPE = 1. When NOTAPE = 0 or 2 (data group B1) this information is provided by the external input file. For example, if NDIM(1) = 0, then the first pollutant has dimensions of mg/l. If NDIM(1) = 1, then the first pollutant has

dimensions of liter^{-1}. This is used when pollutants such as coliforms are routed. When NDIM(1) = 2, pollutant 1 has other concentration dimensions such as JTU, mho, °C, pH, etc.

The entry of pollutant names PNAME(IN,1), PNAME(IN,2), and PNAME(IN,3) and dimension names PUNIT(IN,1), PUNIT(IN,2), and PUNIT(IN,3) is required only when the source of flow and pollutant data is solely data group J1 (NOTAPE = 1, data group B1). If flow and pollutant data are read from an external input file (NOTAPE = 0 or 2, data group B1) this information is already provided.

Naturally, if NP = 2 (data group B1), for example, then the values for IPOLL(3), IPART(3), NDIM(3), PNAME(IN,3), and PUNIT(IN,3) are not required under any circumstances. The order in which this pollutant information is entered determines the numbering of the pollutants.

Data groups E2 through E6 are required only if IPART(IP) = 1 for <u>any</u> pollutant. Refer to Appendix IV for a more detailed discussion of this form of pollutant characterization.

Data Group E2

The variable NVS specifies the manner in which the particles in the waste stream are classified. If NVS = 0, the particles are classified by size and specific gravity. The variable NNR specifies the number of size/specific gravity ranges used to delineate the distribution (up to a maximum of 10 ranges). The size/specific gravity classification remains constant throughout the simulation for each pollutant characterized in this manner. The size and specific gravity ranges are established in data groups E3 and E4. These groups and data group E5 (which enters waste stream temperature data) provide the information with which an average settling velocity is computed for each range (see Appendix IV for details). If NVS =1, the particles are classified by NNR settling velocity ranges specified on data group E3. The average settling velocity for each size range is the average of the range endpoints. Groups E4 and E5 are not required. As with the size/specific gravity ranges, this classification remains constant throughout the simulation for each pollutant characterized by settling velocity. Obviously, a particle size or settling velocity distribution may change as it passes through the storage/treatment plant. This is accomplished by altering the pollutant fractions associated with the various size/specific gravity or settling velocity ranges as they pass through the units. The initial pollutant distributions are entered on the E6 groups. These distributions remain constant, however, as the input to the S/T plant.

The results of a literature review to characterize the pollutants in sanitary sewage, combined sewer overflows, and urban runoff by particle size and specific gravity are shown in Table 7-2. The data presented in the table are not default values and are presented only as a guide in setting up data groups E2 through E6. Local data, through sieve and/or settling column analyses, are always to be preferred.

Data Group E3

Data group E3 is used to enter particle size (if NVS = 0, data group E2) or settling velocity (if NVS = 1, data group E2) ranges. A maximum of ten ranges (as indicated by NNR, data group E2) may be entered. The variables RAN(1,1) and RAN(1,2) through RAN(NNR,1) and RAN(NNR,3) represent the lower and upper bounds of the diameters of settling velocities of particles found in ranges 1 through NNR. The ranges entered in this data group remain constant through a simulation. When NVS = 0 (data group E2), corresponding constant values of specific gravity are read on data group E4.

Table 7-2. Particle Size Distribution

Sanitary Sewage[a]

Particle Size, Microns	Percent Weight in Each Size Range									
	Total Solids					Volatile Solids				
	(1)	(2)	(3)	(4)	(5)	(1)	(2)	(3)	(4)	(5)
< 0.001	69	68	64	64	64	50	42	41	31	46
0.001-1	6	6	7	7	7	9	9	10	14	11
1-100	11	11	11	12	29	18	18	19	24	43
> 100	14	15	18	17		23	25	30	31	

Particle Size, Microns	Percent Weight in Each Size Range			
	Organic Nitrogen			Total P
	(6)	(3)	(7)	(4)
< 0.001	53	22	37	68
0.001-1		11	20	9
1-100	47	34	20	15
> 100		33	23	8

Specific gravity: Suspended solids, 0.80-1.60 (6); Settleable solids, excluding grit, 1.05-1.20 (11); Grit, 2.65 (11)

Combined Sewage[a]

Particle Size, Microns	Percent Weight in Each Size Range
	Suspended Solids
	(8)
< 74	48
74-295	22
295-991	16
991-3327	9
> 3327	5

Specific gravity: Suspended solids, 0.80-2.60 (10); Settleable solids, excluding grit, 1.05-1.20 (11); Grit, 2.65 (11)

Stormwater Runoff (Street Contaminants)[a]

Particle Size, Microns	Percent Weight in Each Size Range
	Suspended Solids
	(9)
< 43	14
43-104	11
104-246	18
246-840	22
840-2000	14
> 2000	21

Specific gravity: Same as reported by reference 11 for combined sewage

[a]Numbers in parentheses refer to the literature cited below:

(1) Rickert and Hunter, 1967
(2) Rickert and Hunter, 1971
(3) Hunter and Heulekekian, 1965
(4) Heulekian and Balmat, 1959
(5) Metcalf and Eddy, Inc., 1972
(6) Helfgott et al., 1970
(7) Painter and Viney, 1959
(8) Envirogenics Co., 1970
(9) Sartor and Boyd, 1972
(10) Dalrymple et al., 1975
(11) Sullivan et al, 1974

Data Group E4

This group is required only if NVS = 0 (data group E2). The variables SPG(1) through SPG(NNR) represent the specific gravity of the particles found in size ranges 1 through NNR (data group E2). Literature values are shown in Table 7-2.

Data Group E5

This group is required only if NVS = 0 (data group E2). The variables TEMP(1) through TEMP(12) represent the average waste-water temperatures for each month of the year. Water temperature has a direct effect on the settling velocity of a particle. This is reflected through the viscosity of the wastewater which is a function of temperature. Refer to Appendix IV for details.

Data Group E6

An E6 data group is required for each pollutant characterized by a particle size/specific gravity or settling velocity distribution (IPART(IP) = 1, data group E1). For example, if IPART(1) = 1 (data group E1) the variables PSD(1,1) through PSD(1,NNR) represent the fraction of pollutant 1 found in the particle size/specific gravity or settling velocity ranges 1 through NNR entering unit 1 (all flows and pollutants must enter the S/T plant at unit 1). Naturally, these fractions should sum to 1.0. Table 7-2 contains some literature values for several pollutants (particle size/specific gravity distributions only).

The distribution entering the S/T plant remains constant throughout the simulation; however, it may change as the pollutants move through the various units. This is an approximation of the more probable situation in which the plant influent distributions change with time. This limitation was necessary due to the fact that no other SWMM blocks generate such distributions for input to the S/T Block. However, the S/T Block can be easily modified to accept time-varying distributions should such a capability be developed in the future.

Storage/Treatment Unit Information
Numbering

Data groups F1 through I1 are repeated for each storage/treatment unit (up to NU, data group B1). The unit number, I, is determined by the order in which each set of data groups F1 through I1 appears. This numbering scheme is important to the permissible configurations of units. There are two rules for numbering units; 1) flows and pollutants exiting from one unit must be directed to a unit with a number greater than its own; and 2) all flows and pollutants entering the S/T system must enter unit 1. The flows entering and exiting a unit are shown in Figure 7-2. Several examples of storage/treatment plant configurations are shown in Figure 7-3.

General Unit Information
Data Group F1

This data group is used to enter the name of unit I.

Data Group F2

The variable IDENT(I) describes the unit I as a non-detention (IDENT(1) = 0) or detention process (IDENT(I) = 1). If IDENT(I) = 1, all or portions of data group H must be included.
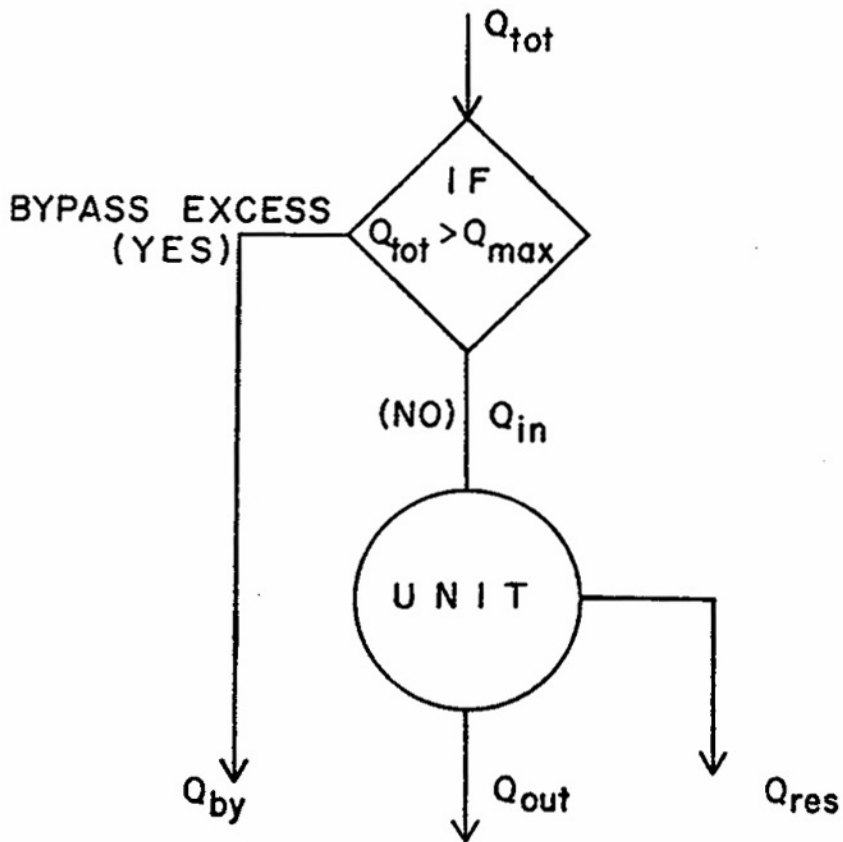
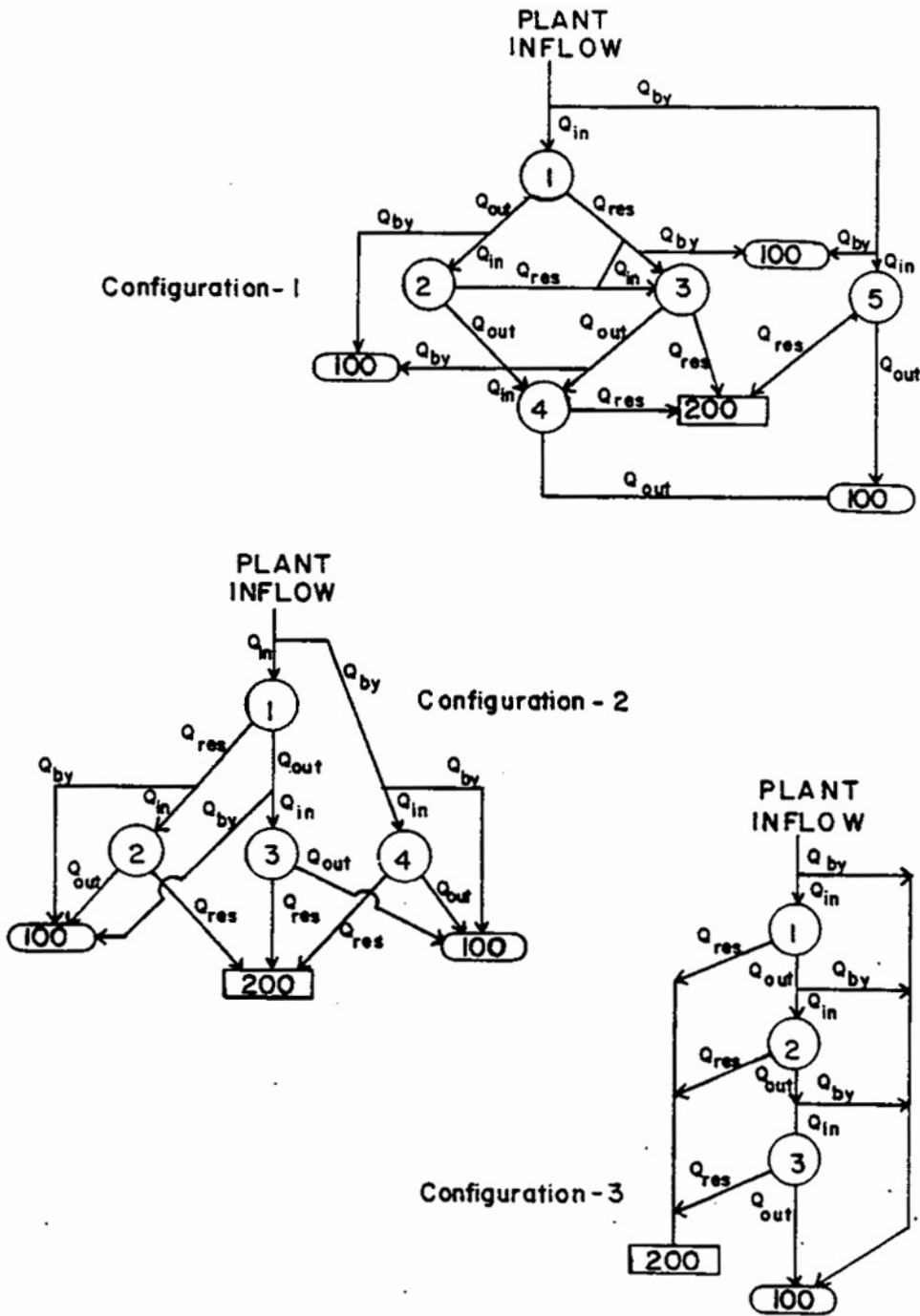Figure 7-2. Flows into, through, and out of a storage/treatment unit.

Figure 7-3. Storage/treatment plant configurations.

Each unit is assigned a maximum inflow, QMAX(I), beyond which all flows and pollutants are bypassed. However, this variable may be set to an abnormally high value for design purposes (i.e., responses at all possible in-puts) or set at a realistic value for modeling existing or proposed facilities. The variable QRF(I) is used to specify the residual flow, as a fraction of the inflow, for non-detention units only (IDENT(I) = 0).

The variables IDIREC(I,1), IDIREC(I,2) and IDIREC(I,3) are used to direct bypassed flow and pollutants, treated outflow, and residuals from unit I to other units. The values entered for these variables represent the unit numbers to which these flows and pollutants are to be directed. Additionally, these flows may be sent directly a downstream block o to ultimate disposal (which simply removes them from the simulation) by specifying IDIREC(I,ID) = 100 or 200, respectively. The flows and pollutants directed to the next block are summed for all the units and transferred as a single stream. Any unit to which flows are directed must have a unit number greater than the source unit (see Figure 7-3 for examples).

Pollutant Removal
Options

Pollutants are removed by settling or obstruction when characterized by particle size/ specific gravity or settling velocity distributions. When they are characterized by magnitude (concentration) alone, removal is simulated through removal equations. Data groups G1 through G3 are used to establish these removal equations. When a particle size/specific gravity or settling velocity distribution is used and the unit is classified as a non-detention process, then a critical size or settling velocity is selected. The model removes all particles with a size or settling velocity greater than or equal to the critical size. Data group G4 is used to enter this parameter.

Data groups G1 through G4 are repeated for each pollutant unless a detention unit is specified (IDENT(1) = 1, data group F2) and a particle size/specific gravity or settling velocity distribution is specified for pollutant IP (IPART(IP) = 1, data group E1). Naturally, these data groups are omitted if no pollutants are routed (NP = 0, data group B1). Again, data groups G1 through G3 are used only if the pollutant is characterized solely by magnitude. Data group G4 is used only if a pollutant is characterized by magnitude and a particle size/specific gravity or settling velocity distribution (IPART(IP) = 1, data group E1) and a non-detention unit is specified (IDENT(I) = 0, data group F2).

Data Group G1

A single flexible functional form is available for use as a pollutant removal equation (see Appendix IV):

$$R = \left(a_{12}e^{a_1 x_1} x_2^{a_2} + a_{13}e^{a_3 x_3} x_4^{a_4} + a_{14}e^{a_5 x_5} x_6^{a_6} + a_{15}e^{(a_7 x_7 + a_8 x_8)} x_9^{a_9} x_{10}^{a_{10}} x_{11}^{a_{11}}\right)^{a_{16}} \qquad (7\text{-}1)$$

where

x_i =   removal equation variables,

a   =   coefficients, and

R   =   removal fraction, $0 \leq R \leq 1.0$.

277

Each removal equation variable, x_i, may represent one of several parameters available in the program at each time step; these options are discussed below (data group G2). With these variables and the coefficients, a_I, the user can develop the desired removal equation. The coefficients are entered on data group G3.

A maximum removal fraction is specified by RMX(I,IP). This is particularly useful for equations which mathematically generate values of the removal fraction, R, that may exceed a reasonable value or 1.0. RMX(I,IP) provides an upper bound on such equations.

Data Group G2

Data group G2 allows the user to assign various program variables to the variables in equation 7-1. For example, if pollutant 1 (IP = 1) is to be removed in unit I by a removal equation, the values given INPUT(I,1,1) through INPUT(I,1,11) assign program variables to the corresponding variables x_1 through x_{11} in equation 7-1. The program variables available for inclusion are shown in Table 7-3. An example removal equation is discussed below.

Data Group G3

The variables A(I,IP,1) through A(I,IP,16) represent the variables a_1 through a_{16} in equation 7-1 as applied to pollutant IP in unit I.

An example of applying equation 7-1 is provided by a suspended solids removal equation used in an earlier version of the Storage/Treatment Block for sedimentation (detention) units:

$$R_{SS} = R_{max}\left(1 - e^{-kt_d}\right) \qquad\qquad (7\text{-}2)$$

where

R_{SS} =   suspended solids removal fraction, $0 \leq R_{SS} \leq R_{max}$,
R_{max} =   maximum removal fraction,
t_d =   detention time, seconds, and
k =   first order decay coefficient, 1/sec.

This equation can be constructed from equation 7-1 by setting a_{12} (or A(I,IP,12)) = R_{max}, a_{13} (or A(I,IP,13)) = $-R_{max}$, a_3 (or A(I,IP,3)) = -k, a_{16} (or A(I,IP,16)) = 1.0, and letting x_3 = detention time, t_d, by setting INPUT(I,IP,3) = 1 (data group G2). All other coefficients, a_j (or A(I,IP,J)), would equal zero. RMX(I,IP) (data group G1) would not be necessary, as R_{max} limits the value of R. Appendix IV contains other examples.

Data Group G4

The variable PSC(I) specifies a critical particle size (if NVS = 0, data group E2) or settling velocity (if NVS = 1, data group E2) that denotes the point above which all particles are removed from the influent. This parameter is included primarily to model such non-detention units as microscreens, fine screens, and coarse screens. An approximation of the removal effectiveness of screens may be obtained by letting PSC(I) equal the aperture size of the screen (see Appendix IV). Data group G4 is required <u>only</u> if IPART(IP) = 1 (data line E1) and IDENT(I) = 0 (data line F2).

Table 7-3. Program Variables Available for Pollutant Removal Equations

Value of INPUT(I,IP,K)[a] (G2 cards)	Non-Detection Units, IDENT(I) = 0 (card F2)	Detection Units, IDENT(I) = 1 (card F2)	
		Perfect plug flow is used, to route pollutants, IROUTE(I) = 0 (card I2)	Complete mixing is used to route pollutants, IROUTE(I) = 1 (card I2)
0	Not used.	Not used.	Not used.
1	Not used.	Detention time of each plug in detention unit I, seconds	Time step size, seconds
2	Concentration of pollutant 1 passing through unit I.[b]	Initial concentration of pollutant 1 in each plug in detection unit I.[b]	Not used.
3	Concentration of pollutant 2 passing through unit I.[b]	Initial concentration of pollutant 2 in each plug in detention unit I.[b]	Not used.
4	Concentration of pollutant 3 passing through unit I.[b]	Initial concentration of pollutant 3 in each plug in detention unit I.[b]	Not used.
5	Removal fraction of pollutant 1 in unit I (used only for pollutants 2 and 3).	Removal fraction of pollutant 1 for each plug in detention unit I (used only for pollutants 2 and 3).	Removal fraction of pollutant 1 in detention unit I (used only for pollutants 2 and 3).
6	Removal fraction of pollutant 2 in unit I (used only for pollutant 3).	Removal fraction of pollutant 2 for each plug in detention unit I (used only for pollutant 3).	Removal fraction of pollutant 1 in detention unit I (used only for pollutant 3).
7	Inflow rate, ft^3/sec [^{3}/sec].	Not used.	Not used.

[a]I = unit number.
 IP = pollutant number.
 K = Subscript of x in equations 7-1.
[b]Dimensions determined by NDIM(IP) on card E1.

Detention Unit Data
Options
Data groups H1 through H8 are used to describe the special characteristics of detention units. Sedimentation, dissolved air flotation, chlorination, and sludge thickening are some of the processes that may be modeled by a detention unit.

These groups primarily describe the hydraulic characteristics of a detention unit and, thus, are required only if IDENT(I) = 1 (data group F2).

Data Group H1
The variable IROUTE(I) specifies the manner in which pollutants are routed in detention unit I. When IROUTE(I) = 0 the unit routes pollutants under the assumption of perfect plug flow. Perfect plug flow is recommended for long, rectangular tanks where settling is the most important removal mechanism and is required when any pollutant is characterized by a particle size/specific gravity or settling velocity distribution (IPART(IP) = 1, data group E1). Removed pollutants are accumulated in plug-flow units, without decay, until removed by the residual flow. When IROUTE(I) = 1 the unit routes pollutants under the assumption of perfect mixing. Complete mixing is most applicable to small tanks where the primary purpose is to thoroughly mix the contents (e.g., rapid-mix chlorination, flocculators, and mixing tanks). Removed pollutant quantities are not allowed to accumulate in completely-mixed units (i.e., no settling). Appendix IV provides further explanation.

The variable IOUT(I) is used to describe the depth-treated outflow relationship that characterizes the discharge of treated outflow (e.g., weir flow) from unit I. There are three options. The first (IOUT(I) = 0) is to provide the model with as many as 16 data pairs describing the depth-outflow relationship (entered in data group I3). The second option (IOUT(I) = 1) is to approximate the relationship by a power equation (entered in data group H4). The third option (IOUT(I) = 2) specifies a constant pumping rate between certain depths (entered in data group H5).

In addition to treated outflow, a residual stream may be drawn from the unit during periods of no inflow or treated outflow. When a residual stream occurs from a plug-flow unit the entire unit contents (including the removed pollutant quantities) are mixed (i.e., the remaining plugs lose their identity) and drawn off until the unit is empty or inflow occurs. If inflow begins before the unit is empty the remaining contents are placed in a single plug for further routing. In a completely-mixed unit, the pollutant concentrations in the residual flow are identical to the concentrations in the treated outflow. Again, the flow is suspended when inflow occurs. The variable IDRAW(I) simply specifies the conditions under which a residual stream begins. If IDRAW(I) = 0, a residual stream is never drawn and the accumulated pollutants (if IROUTE(I) = 0) remain in the unit. If IDRAW(I) ≤ -1, the residuals are drawn off starting at every -IDRAW(I) time steps (but the flow is delayed if inflow and/or treated outflow is in progress). This option corresponds with the situation in which the unit is drained on a regular (e.g., scheduled) basis. If IDRAW(I) ≥ 1, the residuals are drawn after IDRAW(I) time steps of no inflows or treated outflow. The conditions specified by IDRAW(I) ≥ 1 apply directly to the case in which the unit contents are drained after each runoff event.

The variable IRES(I) is used to describe the depth-residual flow relationship that characterizes the draw-off of the residual stream from unit I. If IRES(I) = 0, the user provides the model with as many as 16 data pairs describing the depth-residual flow relationship (entered on

the H3 groups). If IRES(I) = 1, the relationship is approximated by a power equation (entered in data group H6).

Data Group H2
The parameters in this data group are required <u>only</u> when a particle size/specific gravity or settling velocity distribution is used to characterize <u>any</u> pollutant (IPART(IP) = 1, data group E1) and unit O is a plug-flow detention unit (IROUTE(I) = 0, data group H1).
The variable ALEN(I) represents the travel length for plugs in unit I. The variable AMAN(I) is the Manning's roughness coefficient for the surfaces of unit I and is commonly available for many materials. These values are required for the pollutant removal algorithms used when a particle size/specific gravity or settling velocity distribution characterizes a pollutant (see later).

Data Group H3
These groups are used to enter up to 16 sets of data describing the geometry and hydraulics of detention unit I. Each data group enters a value for depth, SDEPTH(I,MM), along with the corresponding values of surface area, SAREA(I,MM), volume, SSTORE(I,MM), treated outflow, SQQOU(I,MM), and residual flow, QQRS(I,MM).
The only required parameters are DEPTH(I,MM) and SAREA(I,MM); the need for the other parameters depends on other factors. If SSTORE(I,MM) is left blank in <u>every</u> H3 data group, the program will estimate the volume at each depth by averaging the surface area at each depth and the lower adjacent depth, multiplying by the difference in depth, and adding the result to the estimated volume at the lower adjacent depth (i.e., by a trapezoidal rule integration). A value for treated outflow, SQQOU(I,MM), is required for each depth only if IOUT(I) = 0 (data group H1). If IOUT = 1 or 2 (data group H1), this relationship is provided by data group H4 or data group H5, respectively. Likewise, a value for residual flow, SQQRS(I,MM), is required for each depth only if IRES(I) = 0 (data group H1). If IRES(I) = 1 (data group H1), this relationship is described by data group H6. The values entered for SQQRS(I,MM) are used during periods in which a residual stream is drawn from the unit (IDRAW(I), data group I1).
The values specified on these groups (or computed as a result of groups H4, H5, and H6) are used to establish relationships between depth, surface area, volume, treated outflow, and residual flow. The program also generates a relationship between depth and the evaporation rate (see data group D1). These relationships are used to route flows through a detention unit using the Puls method (Viessman et al., 1977). This method is described in detail in Appendix IV.

Data Group H4
This data group is required <u>only</u> if a power equation is to describe the depth-treated outflow relationship (IOUT(I) = 1, data group H1). The equation is:

$$Q_{out} = C_1 (D - D_0)^{C_2}$$ (7-3)

where

Q_{out} = treated outflow, ft³/sec [m³/sec],
C_1, C_2 = coefficients,
D = water depth in detention unit, ft [m], and
D_0 = depth below which there is no treated outflow, ft [m].

The user supplies the values of D_0, C_1, and C_2 (program variables D0, C1, and C2, respectively).

Two common outlet structures that may be modeled with equation 7-3 are the orifice and the broad-crested weir. For example, a weir could be modeled by letting $C_1 = 3.33 \diamond L$ where L = length of the weir in feet, $C_2 = 1.5$, and D_0 = depth at the bottom of the weir in feet. These substitutions yield the familiar weir equation.

Data Group H5

This data group is required <u>only</u> if pumping is specified for the treated outflow from a detention unit (IOUT(I) = 2, data group H1). The variables DSTART(I,1) and DSTART(I,2) represent the depths at which the pumping rates QPUMP(I,1) and QPUMP(I,2) begin. The variable DSTOP(I) specifies the depth at which all pumping stops. In other words, the rate QPUMP(I,1) occurs when the depth is equal to or exceeds DSTART(I,1) and the rate QPUMP(I,2) occurs when the depth is greater than or equal to DSTART(I,2). The pumping rate reverts to the rate QPUMP(I,1) when the depth falls below DSTART(I,2) and continues at that rate until the depth falls to DSTOP(I). The value of DSTART(I,1) must be less than or equal to DSTART(I,2) and DSTOP(I) must be less than or equal to DSTART(I,1).

Data Group H6

This data group is required <u>only</u> if a power equation is used to describe the depth-residual flow relationship (IRES(I) = 1, data group H1). The equation is:

$$Q_{res} = C_3 (D - D_1)^{C_4} \qquad (7\text{-}4)$$

where

Q_{res} = residual flow, ft³/sec [m³/sec],
C_3, C_4 = coefficients,
D = water depth in detention unit, ft [m], and
D_1 = depth below which there is no residual flow, ft [m].

The user supplies the values of D_1, C_3, and C_4 (program variables D1, C3, and C4, respectively). Recall that a residual flow occurs only when dictated by IDRAW(I) (data group H1).

Data Group H7

This data group is used to indicate the build up of sludge in a plug-flow detention unit (IROUTE(I) = 0, data group H1). The user specifies the pollutant used to calculate the sludge volume, NPSL(I); the concentration of pollutant NPSL(I) in sludge has accumulated to an unacceptable level. The sludge volume is increased by dividing the amount of pollutant NPSL(I) removed each time step by SLDEN(I). The model assumes that the sludge volume has no effect

on the available storage volume and that no compression occurs. The information on this data group is only used to warn the user of possible maintenance/performance problem.

Data Group H8
    This data group specifies the total volume, WARN(I), and pollutant concentrations, PCO(I,IP), present in the unit at the start of the simulation. Obviously, the values of PCO(I,IP) are not required if $NP = 0$ (data group B1) or $WARN(I) = 0.0$.

Cost Data
Data Group I1
    This data group is required <u>only</u> if $ICOST = 1$ (data group B1).
    The capital cost for each unit is computed as a function of a design flow or volume specified by the user or is calculated by the model as a function of the maximum value recorded during the simulation:

$$C_{cap} = a \, Q_{max}^{b} \qquad\qquad (7\text{-}5)$$

or

$$C_{cap} = a \left(Q_{in}\right)_{max}^{b} \qquad\qquad (7\text{-}6)$$

or

$$C_{cap} = a \, V_{max}^{b} \qquad\qquad (7\text{-}7)$$

or

$$C_{cap} = a \left(V_{obs}\right)_{max}^{b} \qquad\qquad (7\text{-}8)$$

where

C_{cap}	=	initial capital cost, dollars,
Q_{max}	=	maximum allowable inflow, ft^3/sec [m^3/sec],
$(Q_{in})_{max}$	=	maximum inflow encountered during the simulation, ft^3/sec m^3/sec],
V_{max}	=	maximum allowable storage (detention units only), ft^3 [m^3],
$(V_{obs})_{max}$	=	maximum storage encountered during the simulation (detention units only), ft^3 [m^3], and
a, b	=	coefficients (specified by the user).

Equations 7-5 through 7-8 differ only in the variable used to compute the initial capital cost of unit I. The variable KPC(I,1) specifies which variable is used (as shown in Table 7-4) and CC(I,1) and CC(I,2) represent the coefficients a and b.

The operation and maintenance costs are calculated as a function of the variables listed above and the total operating time (calculated as the number of time steps with flow to or from the unit).

$$C_{om} = d\, Q_{nax}^{f} + h D_{op} \qquad\qquad (7\text{-}9)$$

or

$$C_{om} = d\, (Q_{in})_{max}^{f} + h D_{op} \qquad\qquad (7\text{-}10)$$

or

$$C_{om} = d\, V_{max}^{f} + h D_{op} \qquad\qquad (7\text{-}11)$$

or

$$C_{om} = d\, (V_{obs})_{max}^{f} + h D_{op} \qquad\qquad (7\text{-}12)$$

where

C_{om} = operation and maintenance cost, dollars,
D_{op} = total operating time during the simulation period, hours, and
d, f, h = coefficients (supplied by the user).

Equations 7-9 through 7-12 differ only in the variable used to compute the operation and maintenance costs for unit I. The variable KPC(I,2) specified which variable is used and CC(I,3), CC(I,4), and CC(I,5) represent the coefficients d, f, and h.

The user is cautioned not to misinterpret the cost calculated by the model. For example, in a single-event simulation the calculated capital cost can only be considered an estimator of the true capital cost when the simulated event is a design event. Likewise, when operating time is a factor in computing operation and maintenance costs, the calculated costs can be a valid estimator of the true costs only when a long-term simulation is performed. Recent EPA publications provide useful information for the proper selection of the coefficients required in equations 7-5 through 7-12 (EPA, 1976; Benjes, 1976).

Input Flow and Pollutant Data (Group J1)

If NOTAPE = 1 or 2 (data group B1), user-supplied time series input of hydrographs and pollutographs are input in data group J1. The input is in the form of time-flow-concentration(s). Linear interpolation is used to provide intermediate input values if the S/T time step (DS) is not the same as the time interval used in data group J1. (A similar interpolation scheme is used for time series entry from an upstream block via the interface file.) Of course, since linear interpolation is used, the time interval for data input need not be constant. A step function input can be achieved by inputting two flow-concentration values at each time. For instance, assuming DS = 1 hr, the simulation begins at 8:00 a.m., and there are no pollutants, the following J1 entries would result in constant inflows of 5, 8 and 9 cfs over the first three time steps:

```
*      TCAR        QCAR
J1      8.0         5.0
J1      9.0         5.0
J1      9.0         8.0
J1     10.0         8.0
J1     10.0         9.0
J1     11.0         9.0
etc.
```

Pollutant concentration time series are entered only if NP > 0 (data group B1). The pollutant concentrations, PCAR(IP), must be entered in the same order as in data group E1 and have the same dimensions specified by NDIM(IP) and PUNIT(IN,IP) (data group E1). If NOTAPE = 2 (data group B1), the flow and concentrations are added to the values from the external file. All values are instantaneous flows or concentrations (at the end of the time step).

Output

Summary and detailed (time step) output is available for each of up to five units depending on the parameters of data groups C1 and C2. Output may be graphed using the Graph Block only for location 100 (outflow to downstream block), using the interface file. If more than one input location is found on the input interface file, the unaltered time series for thes3 locations will be placed on the output interface file along with that predicted for location 100.

Table 7-4. Storage/Treatment Block Input Data

SWMM INPUT GUIDELINES

There have been many changes made to the input format of SWMM. Following is a short list of the major changes along with explanations and guidelines.

1. Free format input. Input is no longer restricted to fixed columns. Free format has the requirement, however, that at least one space separate each data field.   Free format input also has the following strictures on real, integer, and character data.
    a.   No decimal points are allowed in integer fields. A variable is integer if it has a 0 in the default column. A variable is real if it has a 0.0 in the default column.
    b.   Character data must be enclosed by single quotation marks, including both of the two title lines.
2. Data group identifiers are a requirement and <u>must</u> be entered in columns 1 and 2. These aid the program in line and input error identification and are an aid to the SWMM user. Also blank lines no longer are required to signal the end of sets of data group lines; the data group identifiers are used to identify one data group from another.
3. The data lines may be up to 230 columns long.
4. Input lines can wrap around. For example, a line that requires 10 numbers may have 6 on the first line and 4 on the second line. The FORTRAN READ statement will continue reading until it finds 10 numbers, e.g.,

        Z1 1 2 3 4 5 6
           7 8 9 10

    Notice that the line identifier is <u>not</u> used on the second line.
5. An entry must be made for <u>every</u> parameter in a data group, even if it is not used or zero and even if it is the last required field on a line. Trailing blanks are not assumed to be zero. Rather, the program will continue to search on subsequent lines for the "last" required parameter. Zeros can be used to enter and "mark" unused parameters on a line. This requirement also applies to character data. A set of quotes must be found for each character entry field. For instance, if the two run title lines (data group A1) are to consist of one line followed by a blank line, the entry would be:

        A1 'This is line 1.'
        A1 ''
6. See Section 2 for use of comment lines (indicated by an asterisk in column 1) and additional information.

Variable	Description	Default
	Title: two lines with heading to be printed on output.	
A1	Group identifier	None
TITLE2	Title	Blank
	General Data	
B1	Group identifier	None
NOTAPE	Input data source = 0, Input is from an external input file = 1, Input is supplied in Group J1 = 2, Input is from an external input file and Group J1	0
JNS	External element number from the external block (e.g., JN, Group H1, in Transport Block) which routes flow to the S/T Block. If NOTAPE = 1, the value of JNS is placed on the output file.	None
NDT	Total number of simulation time steps	0
DS	Size of time step, seconds. Required only if NOTAPE = 1	None
NU	Number of storage/treatment units. (Maximum = 5)	1
NP	Number of pollutants routed. (Maximum = 3)	0

Table 7-4.   Continued

Variable	Description	Default
ICOST	Cost calculations performed? = 0, No = 1, Yes	0
METRIC	Metric input-output. = 0, Use U.S. customary units = 1, Use metric units. Metric input indicated in brackets [ ].	0
SAREA	Service area, acres [ha]. Required if NOTAPE = 1.	0.0
	Starting Time and Print Instructions	
C1	Group identifier	None
IDATE	Date at beginning of simulation (6 digit number; year, month, day -- e.g., March 10, 1979 = 790310)	0
TIME	Time at beginning of simulation (24 hour clock, e.g., 5:30 p.m. = 17.5)	0
ISUM	Summary print control parameter = 0, Print a summary at the end of the simulation only = 1, Print an annual summary and a summary at the end of the simulation = 2, Print monthly and annual summaries and a summary at the end of the simulation	0
IDET	Detailed print control parameter = 0, No detailed print of simulation results. > 0, Detailed print of results is provided at every time step that is a multiple of IDET (e.g., IDET = 2, gives a detailed report at every other time step) during specified periods (see below and group C2).	0
NPR	Number of detailed print periods. Up to 8 periods may be specified (See Group C2). Required only if IDET > 0.	0
	Detailed Print Periods	
	NPR (Group C1) periods must be specified. Only date to date periods may be used (e.g., 790720 to 790806). Required only if IDET > 0 (Group C1).	
C2	Group identifier	None
ISTART(1)	First detailed print period starting date (e.g., July 20, 1979 = 790720).	None
IEND(1)	First detailed print period ending date (e.g., August 6, 1979 = 790806).	None
■	Repeat for second period, etc. up to NPR Group C1 periods - place all ISTART and IEND pairs on a single line.	
ISTART(NPR)	Last detailed print period starting date.	
IEND(NPR)	Last detailed print period ending date.	None
	Evaporation Data	
	Evaporation data. Required only if there are detention units. (IDENT(I) = 1 for some units, see Group F2).	
D1	Group identifier	None
E(1)	Evaporation rate, January in./day [mm/day]	0.0
■	■	
E(12)	Evaporation rate, December in./day [mm/day]	0.0

Table 7-4. Continued

Variable	Description	Default
	Groups E1 - E6: Pollutant Characterization	
	REQUIRE GROUPS E1 - E6 ONLY IF NP > 0 (GROUP B1)	
	General Pollutant Characteristics	
E1	Group identifier	None
IPOLL(1)	Pollutant 1 selector. Required only if NOTAPE = 0 or 2 (Group B1). The value selected depends on the order in which the pollutants were placed on the external input file. For example, if suspended solids was the third pollutant listed on the file and it was desired for use in the S/T block, then IPOLL(1) = 3.	None
NDIMI(1)	Dimensions for pollutant 1. Required only if NOTAPE = 1 (Group B) = 0, Dimensions are mg/l = 1, Dimensions are liter (-1) = 2, Other concentration dimensions are used. (e.g., JTU, deg. C. pH)	0
IPART(1)	Particle size/specific gravity or settling velocity distribution parameter = 0, Distribution not used to characterize pollutant 1. = 1, Distribution used to characterize pollutant 1	0
PNAME1(IN,1)	Pollutant 1 name. Required only if NOTAPE = 1 (Group B1)	None
PUNIT1(IN,1)	Pollutant 1 dimension label. Required only if NOTAPE = 1 (Group B1)	None
IPOLL(2)	Pollutant 2 selector. Required only if NP ≥ 2 and NOTAPE = 0 or 2 (Group B1). See above.	None
NDIMI(2)	Dimensions for pollutant 2. Required only if NP ≥ 2 and NOTAPE = 1 (Group B1). See above.	0
IPART(2)	Particle size/specific gravity or settling velocity distribution parameter. Required only if NP ≥ 2 (Group B1) See above.	0
PNAMEI(IN,2)	Pollutant 2 name. Required only if NP ≥ 2 and NOTAPE = 1. (Group B1)	0
PUNITI(IN,2)	Pollutant 2 dimension label. Required only if NP ≥ 2 and NOTAPE = 1. (Group B1)	None
IPOLL(3)	Pollutant 3 selector. Required only if NP = 3 and NOTAPE = 0 or 2 (Group B1) See above.	0
NDIMI(3)	Dimensions for pollutant 3. Required only if NP = 3 and NOTAPE = 1 (Group B1). See above.	0
IPART(3)	Particle size/specific gravity or settling velocity distribution parameter. Required only if NP = 3. (Group B1). See above.	0
PNAMEI(IN,3)	Pollutant 3 name. Required only if NP = 3 and NOTAPE = 1 (Group B1)	None
PUNITI(IN,3)	Pollutant 3 dimension label. Required only if NP = 3 and NOTAPE = 1 (Group B1)	None
	GROUPS E2 - E6 ARE REQUIRED ONLY IF IPART (IP) = 1 (GROUP E1) FOR ANY POLLUTANTS.	
E2	Group identifier	None
NVS	Classification parameter = 0, Particle size/specific gravity distribution is used to classify particles in waste stream. = 1, Settling velocity distribution is used	0
NNR	Number of particle size ranges or settling velocities used to classify particles in waste stream (Max of 10).	None

Table 7-4. Continued

Variable	Description	Default
	Particle size (if NVS = 0, group E20 or settling velocity (if NVS = 1, group E2) range data.	
E3	Group identifier	None
RAN(1,1)	Lower bound of size or velocity range 1, microns or ft/sec [cm/sec]	None
RAN(1,2)	Upper bound of size or velocity range 1, microns or ft/sec [cm/sec]	None
RAN(2,1)	Lower bound of size or velocity range 2, microns or ft/sec [cm/sec]	None
RAN(2,2)	Upper bound of size or velocity range 2, microns or ft/sec [cm/sec]	None
■	■	
	Repeat for each size or velocity range, up to NNR (Group E2) ranges.	
	Specific gravity data.	
	Required only if NVS = 0 (Group E2).	
E4	Group identifier	None
SPG(1)	Specific gravity for particles in size range 1.	None
SPG(2)	Specific gravity for particles in size range 2.	None
■	■	
	Repeat for each size range, up to NNR (Group E2) ranges.	
	Waste Stream Temperature Data	
	* * * Required only if NVS = 0 (Group E2). * * *	
E5	Group identifier	None
TEMP(1)	Waste stream temperature, January ^{o}F. [^{o}C]	None
■	■	
TEMP(12)	Waste stream temperature, December ^{o}F. [^{o}C]	None
	Fraction of pollutant associated with each particle size/specific gravity or settling velocity range (Group E3). Repeat these lines for each pollutant for which IPART(IP) = 1 (Group E1).	
E6	Group identifier	None
PSD(IP<1)	Fraction of pollutant IP in range 1.	None
PSD(IP,2)	Fraction of pollutant IP in range 2.	None
■	■	
	Repeat for each range up to NNR (Group E2) ranges.	
	REPEAT GROUPS F1 - I1 FOR EACH UNIT I. THERE WILL BE NU (GROUP B1) SETS. THE UNIT NUMBER IS DICTATED BY THE ORDER IN WHICH THE SETS OF GROUPS F1 - I1 ARE READ.	
	Unit Name	
F1	Group identifier	None
UNAME(I,ID)	Name of unit	None

Table 7-4. Continued

Variable	Description	Default
	General Unit Parameters and Flow Directions	
F2	Group identifier	None
IDENT(1)	Detention modeling parameter	0
	= 0, Unit is the non-detention type	
	= 1, Unit is the detention type	
QMAX(I)	Maximum inflow (above which bypass occurs), cfs [cu.m/sec].	None
QRF(I)	Residual flow as a fraction of the inflow. Required only if IDENT(I) = 0. Residual flows for detention units (IDENT(I) = 1) are determined in Groups H1, H3, and H6.	None
IDIREC(I,1)	Unit number to which bypass is directed (must be greater than 1)	None
	= 1 - 5, Downstream S/T unit	
	= 100, Next block	
	= 200, Ultimate disposal	
IDIREC(I,2)	Unit number to which treated outflow is directed (must be greater than 1). See above.	None
IDIREC(I,3)	Unit number to which residuals stream is directed (Must be greater than 1). See above. If IDRAW(I) = 0 (Group H1), set equal to any number > 1.	None
	Data Groups G1 - G4: Pollutant Removal	
	REQUIRED ONLY IF NP > 0 (GROUP B1)	
	REPEAT GROUPS G1 - G3 FOR EACH POLLUTANT FOR WHICH IPART(IP) = 0	
G1	Group identifier	None
RMX(I,IP)	Maximum removal fraction (≤ 1.0)	None
	Removal Equation Variable Group (Equation 7-1)	
	Requires two input lines.	
G2	Group identifier	None
INPUT(I,IP,1)	Program variable for equation variable x_1.	0
	= 0, Not used.	
	For values = 1 - 7, see Table 7-3 in text.	
INPUT(I,IP,2)	Program variable for equation variable x_2. See above.	0
■	■	
	Repeat for each program variable x_i.	
	Equation 7-1 Coefficients	
	The coefficients must be consistent with the units used (see METRIC Group B1).	
G3	Group identifier	None
A(I,IP,1)	Value of coefficient a_1	0.0
A(I,IP,2)	Value of coefficient a_2	0.0
■	■	
	Repeat for each coefficient a_j.	

Table 7-4. Continued

Variable	Description	Default
	Critical Particle Size or Settling Velocity	
	Required only if IPART(IP) = 1 (Group E1) for <u>any</u> pollutant and unit I is a non-detention unit, IDENT(I) = 0 (Group F2)	
G4	Group identifier	None
PSC(I)	Critical particle size, microns, (if NVS = 0, Group E2) or settling velocity, ft/sec [cm/sec], (if NVS = 1, Group E2)	None
	Data Groups H1 - H8: Detention Unit Data	
	Required only if IDENT(I) = 1 (Group F2)	
	General Detention Unit Parameters	
H1	Group identifier	None
IROUTE(I)	Pollutant routing parameter = 0, Plug flow mode is used. = 1, Complete mixing mode is used. (Note: Particle size or settling velocity distribution are not routed through completely-mixed units.)	0
IOUT(I)	Treated outflow routing parameter = 0, The depth-treated outflow relationship is described by as many as 16 data pairs in Group H3. = 1, The depth-treated outflow relationship is described by a power equation in Group H4. = 2, The depth-treated outflow relationship is controlled by the pumps described in Group H5.	0
IDRAW(I)	Residuals stream draw-off scheme ≤-1, A residual stream is drawn off starting at every -IDRAW(I) time step (if possible). = 0, Residuals are never drawn off. ≥ 1, A residuals stream (if available) is drawn off only after IDRAW(I) time steps of no inflow or treated outflow.	0
IRES(I)	Residual stream routing parameter Required only if IDRAW(I) not = 0 = 0, The depth-residual flow relationship is described by as many as 16 data pairs in Group H3. = 1, The depth-residual flow relationship is described by a power equation in Group H6.	0
	Detention unit (plug flow only) parameters, required when pollutants are characterized by a particle size/specific gravity or settling velocity distribution. Thus, this line is required only if IPART(IP) = 1, for any pollutant (Group E1) and IROUTE(i) = 0 (Group E2)	
H2	Group identifier	None
ALEN(I)	Travel length for plug flow, ft [m]	0
AMAN(I)	Manning's roughness coefficent for detention unit surfaces.	None

Table 7-4. Continued

Variable	Description	Default
	Data for Sets of Depth, Surface Area, Volume, Treated Outflow and Residual Flow	
	Each line contains a column for a unit depth and the corresponding values of area, volume, treated outflow, and residual flow. The columns for treated outflow and residual flow may be left blank depending on the values of IOUT(I) and IRES(I) in Group H1. If no values for volume are entered, the program estimates volume from the depth-surface area relationship. Order the data from the bottom of the unit (SDEPTH(I,1) = 0.0) to the maximum depth (including freeboard). There may be as many as 16 lines.	
H3	Group identifier	None
SDEPTH(I,MM)	A unit depth, ft [m]	None
SAREA(I,MM)	Surface area corresponding to the above depth, sq. ft [sq. m]	None
SSTORE(I,MM)	Volume corresponding to the above depth cu. ft [cu. m]	None
SQQOU(I,MM)	Treated outflow at the above depth cfs [cu.m/sec]	None
SQQRS(I,MM)	Residuals stream flow at the above depth cfs [cu.m/sec]. Occurs only when IDRAW(I) (Group H1) permits.	None
	Depth vs. Treated Outflow Power Equation (Equation 7-3)	
	Required only if IOUT(I) = 1 (Group H1). Coefficients must be consistent with the units used. (See METRIC, Group B1.)	
H4	Group identifier	None
C1	Depth-treated outflow equation coefficient, C_1	0.0
D0	Depth below which no treated outflow occurs, D_0	0.0
C2	Depth-treated outflow equation coefficient, C_2	0.0
	Treated Outflow Pumping	
	Required only if IOUT(I) = 2 (Group H1)	
H5	Group identifier	None
DSTART(I,1)	Depth at which pumping rate QPUMP(I,1) begins, ft [m]	None
DSTART(I,2)	Depth at which pumping rate QPUMP(I,2) begins, ft [m]. Must be $\geq$ DSTART(I,1)	None
QPUMP(I,1)	Pumping rate when depth $\geq$ DSTART(I,1) cfs [cu.m/sec]	None
QPUMP(I,2)	Pumping rate when depth $\geq$ DSTART(I,2) cfs [cu.m/sec]	None
DSTOP(I)	Depth below which all pumping stops, ft [m]. Must be $\leq$ DSTART(I,1).	None
	Depth vs. Residual Flow Power Equation (Equation 7-4)	
	Required only if IRES(I) = 1 (Group H1). Coefficients must be consistent with the units used (see METRIC, Group B1).	
H6	Group identifier	None
C3	Depth-residual flow equation coefficient C_3	0.0
D1	Depth below which no residual flow occurs D_1	0.0
C4	Depth-residual flow equation coefficient C_4	0.0

Table 7-4. Continued

Variable	Description	Default
	Sludge Generation in Unit I	
	Required only if I is a plug-flow detention unit (IROUTE(I) = 0, Group H1) and NP > 0 (Group B1).	
H7	Group identifier	None
NPSL(I)	Pollutant responsible for sludge generation. Required only if a sludge depth warning message is desired = 0, Not used. = 1, 2, or 3, Pollutant used to generate sludge volume (must correspond to the position in Group E1).	0
SLDEN(I)	Concentration of pollutant NPSL(I) in sludge. Required only if NPSL(I) ≥ 1. The dimensions used must be consistent with those indicated by NDIM(I) (Group E1).	None
SLDMAX(I)	Maximum sludge depth, ft [m]. A warning message is printed if this depth is exceeded by the accumulated sludge. Required only if NPSL(I) ≥ 1.	None
	Initial Conditions in Detention Unit I	
H8	Group identifier	None
WARN(I)	Total volume of water in unit at the start of the simulation, cu.ft [cu.m]	0.0
	The following concentrations must be given with dimensions consistent with those entered in Group E1 (NDIM(IP)) if NOTAPE = 1 (Group B1) or on the external input file if NOTAPE = 2.	
PCO(I,1)	Concentration of pollutant 1 in the unit at the start of the simulation. Required only if NP ≥ 1 (Group B1) and WARN(I) > 0.0.	0.0
PCO(I,2)	Concentration of pollutant 2 in the unit volume at the start of the start of the simulation. Required only if NP ≥ 2 (Group B1) and WARN(I) > 0.0	0.0
PCO(I,3)	Concentration of pollutant 3 in the unit volume at the start of the start of the simulation. Required only if NP = 3 (Group B1) and WARN(I) > 0.0	0.0
	REQUIRED ONLY IF ICOST = 1 (GROUP B1)	
	Cost Data for Equations 7-5 to 7-8 (Capital Costs) and Equations 7-9 to 7-12 (Operation and Maintenance Costs)	
	The coefficients must be consistent with the units used (see METRIC, Group B1).	
I1	Group identifier	None
KPC(I,1)	Type of cost variable used in calculating initial capital cost. = 0, Not used. = 1, Maximum allowable inflow, QMAX(I), cfs [cu.m/sec] is used. = 2, Maximum inflow observed during simulation, QMAXS(I), cfs [cu.m/sec], is used. = 3, Maximum allowable storage, VMAX(I) cu.ft [cu.m] is used. (Not applicable if IDENT(I) = 0, Group F2) = 4, Maximum storage observed during simulation, VMAXS(I), cu.ft. [cu.m] is used (Not applicable if IDENT(I) = 0 (Group F2)	0
CC(I,1)	Initial capital cost equation coefficient, a.	0.0
CC(I,2)	Initial capital cost equation coefficient, b.	0.0
KPC(I,2)	Type of cost variable used in calculating operation and maintenance costs. See list for initial capital cost (above).	0

Table 7-4. Continued

Variable	Description	Default
CC(I,3)	Operation and maintenance costs equation coefficient, d	0.0
CC(I,4)	Operation and maintenance costs equation coefficient, f	0.0
CC(I,5)	Operation and maintenance costs equation coefficient, h	0.0

<div align="center">

Input Flow and Pollutant Time Series
REQUIRED ONLY IF NOTAPE = 1 OR 2 (GROUP B1)

</div>

All flows and concentrations are instantaneous values at the indicated time. A constant time interval is not required; linear interpolation is used to obtain intermediate values of flow and concentrations. Hence, the difference between two time entries, TCAR, should not be less than the time step, DS, unless a step function input is desired.

The concentration units must be identical to those in Group E1 (NDIM(IP)) if NOTAPE = 1, Group B1, or on the external input file if NOTAPE = 2.

Variable	Description	Default
J1	Group identifier	None
TCAR	Time of day, decimal hours, e.g., 6:30 p.m. = 18.5. If the first TCAR value is ≤ initial time, the program will read succeeding J1 data groups until TCAR ≥ initial time. If simulation goes beyond one day, i.e., times > 24.0, then continue with times greater than 24. I.e., TCAR should not be reset at beginning of day.	0.0
QCAR	Flow entering S/T plant (at unit 1), cfs [m^3/sec].	0.0
PCAR(1)	Concentration of pollutant 1 entering S/T plant (at unit 1). Required only if NP ≥ 1 (Group B1) and QCAR > 0.0	0.0
PCAR(2)	Concentration of pollutant 2 entering S/T plant (at unit 1). Required only if NP ≥ 2 (Group B1) and QCAR > 0.0	0.0
PCAR(3)	Concentration of pollutant 3 entering S/T plant (at unit 1). Required only if NP = 3 (Group B1) and QCAR > 0.0	0.0

<div align="center">

END OF STORAGE/TREATMENT BLOCK INPUT DATA

At this point, the program seeks new input from the Executive Block.

</div>