CIVILDESIGN®

HYDROLOGY/HYDRAULICS

Operators Manual

JOSEPH E. BONADIMAN AND ASSOCIATES, INC.

*** REGISTERED TRADEMARKS ***

CIVILDESIGN and CIVILCADD are *Registered Trademarks* and are the exclusive properties of Joseph E. Bonadiman and Associates, Inc.

** DISCLAIMER **

Every reasonable effort has been made to assure that the results obtained from this CIVILDESIGNB/CIVILCADD® software are correct; however, Joseph E. Bonadiman and Associates, Inc. assumes no responsibility for any results or any use made of the results obtained by using these programs.

*** LIMITED WARRANTY ***

Even though Joseph E. Bonadiman and Associates, Inc. has tested the CIVILDESIGN® CIVILCADD® software and reviewed this document, Joseph E. Bonadiman and Associates, Inc. makes no warranty or representation, either exposed or implied, with respect to the CIVILDESIGN® CIVILCADD® software, its quality, performance, merchantability, or fitness for a particular purpose.

In no event will Joseph E. Bonadiman and Associates, Inc. be liable for direct, indirect, special, incidental, or consequential damages resulting from the use, or misuse, of the CIVILDESIGN® software, or for any real or discerned defects in the software or its documentation, even if advised of the possibility of such damages. In particular, Joseph E. Bonadiman and Associates shall have no liability for any programs, or data stored or used with, or any products thereof produced by the *CIVILDESIGN* software.

In particular, Joseph E. Bonadiman and Associates, and / or CivilDesign Corportation, shall have no liability for any programs, or data stored or used with, or any products thereof produced by the *CIVILDESIGN*(R) software.

The warranty and remedies set forth above are exclusive and in lieu of all others, oral or written, express or implied. No agent or representative of Joseph E. Bonadiman and Associates, Inc. is authorized to make any modification, extension, or addition to this warranty.

*** DOCUMENTATION ***

Overview

Hydrology/Hydraulics Menu

These programs are used to design open and closed channel structures and to perform hydrology calculations and analyses. Each program level is briefly described below, and in greater detail in subsequent sections of this manual.

See *Appendix A* for initial diskette loading instructions, if you have not already loaded the programs onto your *PC.*

Enter CIVILD to access the *Hydrology/Hydraulics Menu* shown below.

```
        CIVILCADD/CIVILDESIGN            Engineering.Software

    HYDROLOGY/HYDRAULICS  PROGRAMS:

      The following programs are available:

     1 - Rational Hydrology method programs.
      2 - Unit Hydrograph method programs.
      3 - Flood Hydrograph routing.
    4 - Water Surface Pressure Gradi ent (WSPG).
      5 - General Hydraulics (Irregular channel, Trapezoidal;
           Box, Pipe, Wier, Street, Street Inlet; Pump, Turbine):
      6 - Single pipe flow, pressure/nonpressure.
      7 - Sanitary Sewer Network Design/Analysis.
      8 - None, Exit program.

                    Enter program option desired >
```

Menu item **#1** is used to calculate storm run-off using the *Rational* Method. *CIVILDESIGN* Rational Method programs currently available include the Universal Method for use in any geographical area, and specific programs for use in San Bernardino, Riverside, San Diego, Kern (and City of Bakersfield), and Orange Counties, California. Also available is the Los Angeles and Ventura Counties rational method storm run-off programs for areas of 100 acres or less, and the *Los Angeles County Modified Rational* (F0601) program. See Section 1 for more information.

Menu item **#2** is used to calculate storm run-off using the *Unit-Hydrograph* Method. *CIViLDESIGN* Unit-Hydrograph programs currently available include the Universal Method for use in any geographical area, and specific programs for use in San Bernardino, Riverside, Orange, Kern, Los Angeles, and San Diego Counties, California. See Section 2 for more information.

Menu item **#3** is used to perform storm run-off routing calculations, and is designed to assist the engineer in designing or evaluating channels, retarding basins, flow-by basins, or computing and displaying the resultant hydrograph after routing it through a channel, or combining the resulting hydrograph with another hydrograph. For Southern California users, this level also includes a routing program for Los Angeles County that models retarding basins using a ***FO601 Hydrograph*** file. See Section 3 for more information.

Menu item #4 is used to access the Los Angeles County Water Surface Pressure Gradient programs. The original *main frame* programs ***are public domain*** programs from which these ***PC CIVILDESIGN*** versions were developed. We have written input and edit routines that allow you to enter and edit data without having to consider the specific ***main frame*** card format column locations for each data element. A ***Help*** routine has also be added to assist you. See Section 4 for more information on the L.A. County WSPG Programs.

Menu item **#5** is used to calculate either the flow capacity or the amount of flow in Irregular-shaped, Trapezoidal, and Box channels, in Pipes, and through Weir structures. Program can also calculate flow rates for up to 10 channel structures in a system, in either pressure or non-pressure flow, through a range of up to 100 ***depth steps.*** Program also analyzes street flow, with or without street inlets, and analyzes Pump/Turbines. See Section 5 for more information.

Menu item #6 is used to quickly evaluate a single pipe under pressure or nonpressure conditions using various types of conditions. See Section ***6*** for more information.

Menu item #7 is used to design a new sanitary sewer system or to analyze flows in an existing system. In the ***design*** mode, the program calculates and tabulates the flow in each line and the total flow in the system, and calculates pipe sizes, slopes, and invert elevations. In the *anaylis* mode, it evaluates an existing system, determining pipe capacities (depth of flow) throughout the system. See Section 7 for more information.

Overview of Hydrology Programs

The present ***CIVILDESIGN*** hydrology program package consists of Rational and Unit Hydrology programs, the HEC1 single event hydrology program, and hydraulic programs such as HEC2 for open channel flow and the L.A. County Water Surface Pressure Gradient (WSPG) program for any type of open channel or closed channel (pipe or box) flow. To augment the unit hydrograph programs, a flood hydrograph routing program is available for the design of retarding basins, flowby basins, channel routing, and combining hydrographs.

Note: The Army Corps of Engineers ***HEC*** programs are not included in the above menu, since they must be stored in separate directories on your system; however, they are available from ***CIVILDESIGN, or free, from http://www.wrc-hec.usace.army.mil.***

RATIONAL You will be required to enter a study or file name to start the program, and then the initial control data (such as rainfall data for the rational programs). The basic program options are to Build (or create) the file, Run, Correct or Add data to the file.

Building a file: When building or creating the file, the program gives immediate answers to the operation accomplished, and then gives you the choice of rejecting or accepting the results. If you accept the results, the program adds this data to the input file and returns you to the operation menu. At any time in this process you may return to the main menu by typing

[R], or to the previous screen display by typing [B] for back, or [T] for top of screen.

Correct a file: The Correct (or Edit) File option will first review the control data. It will then display a list of the operations used or entered in the input file. You may select one item from this list at a time and either correct the item, delete the item, or add a different operation above or below the number selected. If an operation is being corrected, the program will read the data entered and display these values (dim display) by the questions. If the value displayed is correct, press the RETURN [RTN] key to use it. If the value must be changed, enter the new value, then press RETURN [RTN] and the old value will be over-written.

NOTE: The unit hydrology and the WSPG programs use different methods for editing data. In the unit hydrograph programs, you review the entire file starting from the beginning. At any point in the review you may return to the main menu, and any parameters changed will also be changed in the input file.

Add to a file: The Add option in the rational and routing programs scans to the end of the input data file to update the program with the results to that point. Then, you may proceed *building* the file with immediate answers to options shown on the screen as in the *build* mode.

NOTE: There is no add option in the Unit Hydrology programs. The input file MUST BE COMPLETED to the end in order to run the program. Partially completed unit hydrograph input files (user exit before completion), may be completed by using the Correct option.

Run a file: The Run option in all programs will run the input data file and output the results either to the screen, to a user designated output file, or directly to the default printer. When displaying the file to the screen, you must alternately use Ctrl-S to stop the display, or Ctrl-Q to start the display to review all of the results. When sending the results to the default printer, a *STANDBY* will appear on the screen while the program is writing a temporary output file. When creating an output file for later printing, some of the programs advise against using the SAME file name as the input file; other programs will name the file with the input file name and an ".OUT" extension. After creating the output file, you MUST EXIT to system level to view (VUE), print, or *type* the output file.

Los Angeles/Ventura Counties Rational Method

The above *OVERVIEW of HYDROLOGY PROGRAMS* does not necessarily apply to the *Los Angeles County* and *Ventura County Rational Method* programs. *Help files* are included with these programs, which can be printed for permanent reference. See Section 1 for additional information.

(This page intentionally left blank)

Section 1

Rational Hydrology Programs

When [1] is entered from the *Hydrology/Hydraulics Menu (see Overview),* the CRT will display the menu shown below.

```
CIVILCADD/CIVILDESIGN Engineering Software

RATIONAL METHOD HYDROLOGY PROGRAMS:

The following programs are available:

 1 - Universal (Includes SI unit option).
 2 - San Bernardino County.
 3 - Riverside County.
 4 - Orange County.
 5 - San Diego County (and City of San Diego).
 6 - City of Chula Vista, San Diego County.
 7 - LA County ( less than 100 ac. ).
 8 - LA County Modified Rational Method (F0601).
 9 - Ventura County (less than 100 ac.).
10 - Kern County (and City of Bakersfield).
11 - None, Exit program.-

          Enter program option desired >
```

Select number, then enter the menu number that corresponds to your requirement.

Data Required:

Generally, all rational programs require rainfall, soil type, type of development, and topographic data for the area under study.

RAINFALL: This data is included in the Orange and Riverside County programs; however, all of the other rational method programs require you to enter this data from rainfall maps for your specific area. The Universal Rational program allows two methods of entering rainfall data. You can enter rain-intensity data pairs starting from 5 minutes, up to approximately 180 minutes (or the maximum time of concentration used); or you can enter the rainfall and year (2 pairs required if the study year is not the same as the rainfall year), and a log slope of the rainfall intensity-time line relationship.

SOIL DATA: You will enter a type of development, i.e., 1/4 acre lots, along with the soil type (A, B, C, or D; where A = sand and D = clay). The programs will compute a rainfall soil loss rate(in/hr). Most of the rational programs also allow you to manually enter the soil data with various options. The program then computes the soil loss rate.

TOPOGRAPHIC DATA: You should obtain a topographic map of the study area and delineate the tributary drainage subareas. Determine the area in acres of each of these subareas, starting at the top of each stream, with an initial area not larger than 10 Acres or longer than 1000 feet of stream flow. The elevations of the top and bottom of each subarea and stream points should be marked, along with where the streams confluence (join each other).

Program Operations:

Upon accessing the selected Rational Hydrology program, the CRT will display the *Main Menu* shown on the next page (Note: some programs differ slightly from this display).

```
Rational Hydrology Program Options:
     1 - Create a new study file
     2 - Run file, detailed report
     3 - Run file, form report (132 characters wide)
     4 - Change file entries
     5 - Add to study file
     6 - Print listing of study file entries (no results)
     7 - None of the above, exit program

Enter program option desired
```

When any of the above items are selected, you will be asked for the study *NAME* (up to 6 characters). Each of the above items is described below.

Create a New Study File

You will be asked to enter the ***control data,*** i.e., the rainfall data, and other general criteria, parameters, and options that you want the program to use. Note: The Riverside and Orange County programs have the rainfall data ***built into*** the program.

> Note: If your study involves streets and storm drains, you should use the ***storm event* year** that your approving agency requires for maintaining street flow within top-of-curb (normally a 10 year storm). **By** doing this, you can properly design the storm drain and street inlet sizes to carry the flow within top-of-curb and then check the designed system for maximum flow rate conditions (100 year storm).

After the ***control data*** is entered, the **CRT** will display:

```
1 - INITIAL subarea input, top of stream
2 - STREET flow thru subarea, includes subarea runoff
3 - ADDITION of runoff from subarea to stream
4 - STREET INLET + parallel street & pipe flow + area
5 - PIPEFLOW travel time (program estimated pipesize) **
6 - PIPEFLOW travel time (user specified pipesize)
7 - IMPROVED channel travel time (open or box) **
8 - IRREGULAR channel travel time **
9 - USER specified entry of data at a point
10 - CONFLUENCE at downstream point in CURRENT stream
11 - CONFLUENCE of MAIN streams
     **NOTE: These options with do not include subarea runoff
Enter the desired subarea option
```

Note: Only items 1 and **9** will appear in the above menu when the current stream flow rate is zero.

Initial Subarea Input, Top of Stream:

You must start computations for a stream with an *INITIAL AREA* or *USER INPUT of DATA* at a point. Normally, the *INITIAL AREA* option is used. However, if you are starting with a stream that has a known flow rate into the stream area, the *USER INPUT* option would be used. The *INITIAL* AREA option calculates the time of concentration for the outlet of the initial area and the corresponding flow rate. In most cases the **initial area should be less than 10 acres and have a flow distance that is less than 1000 feet long - Orange County is less.**

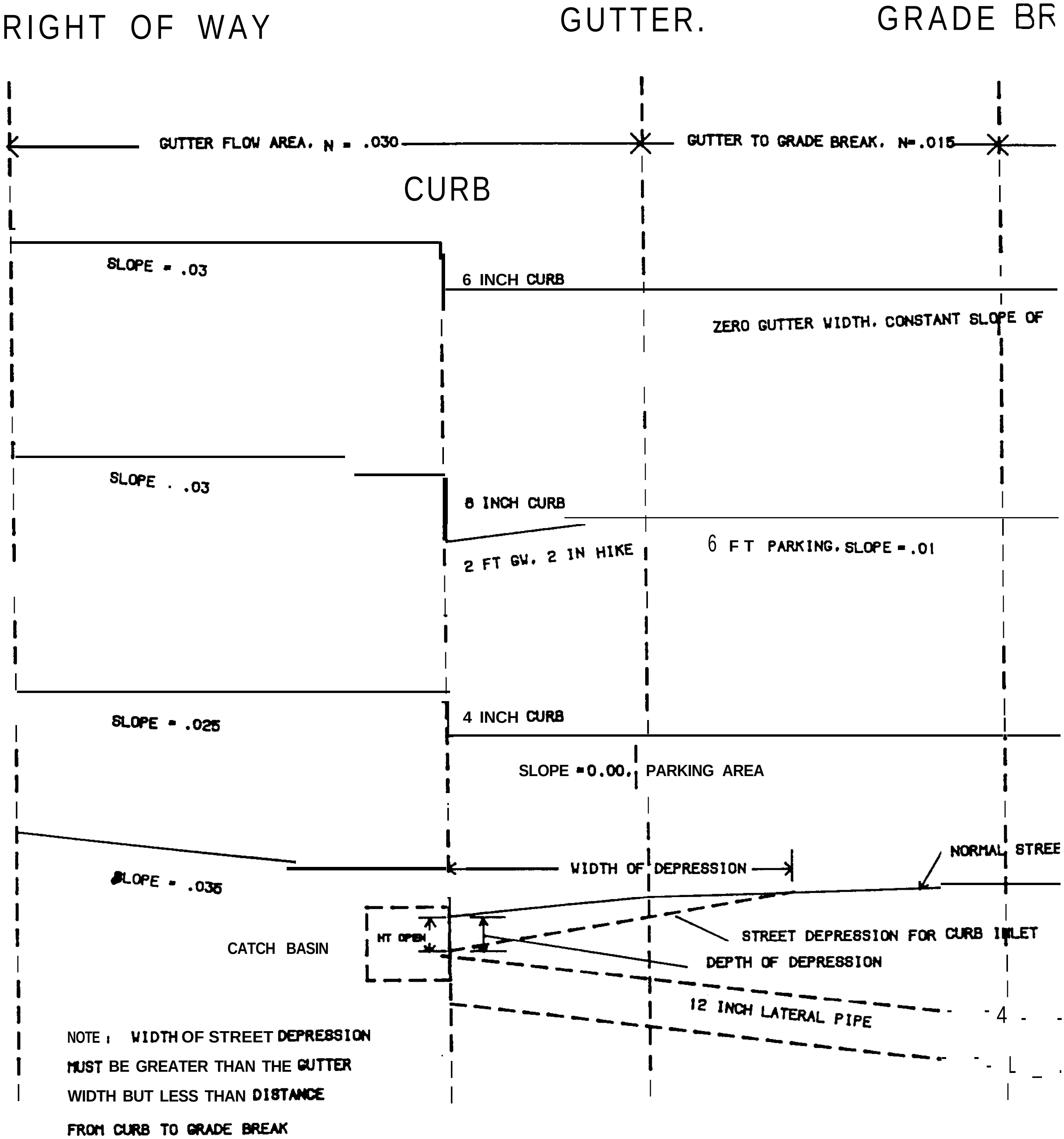
After the *Initial Subarea* data is entered, any of the above menu items **2** through **8** can be selected and used. Each is explained below.

Street Flow:

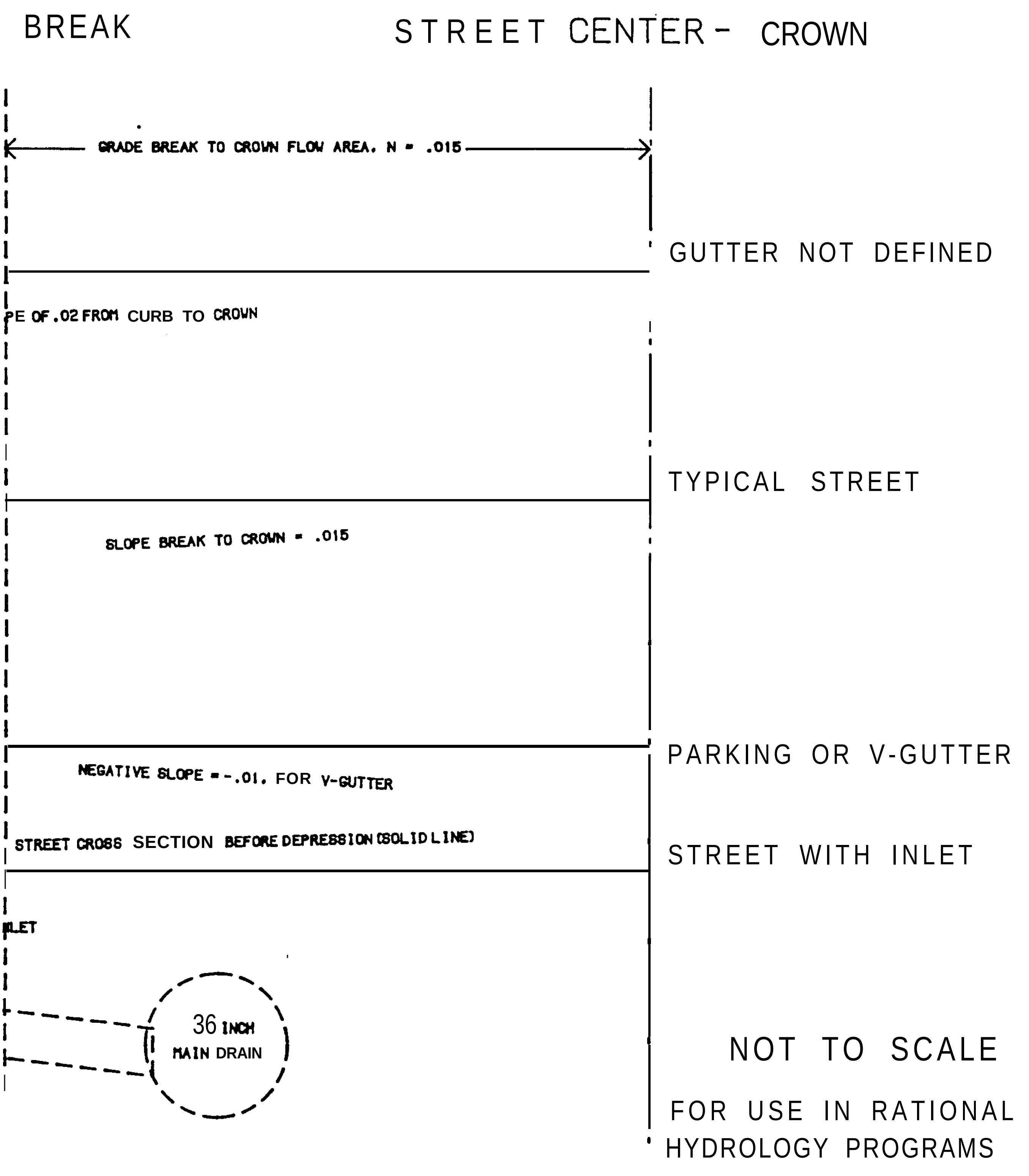
With this option, the assumption is that in a developed area the stream is allowed to flow down a street until the street is flowing full (up to the top of curb, or to the right-of-way line in 100 year storm events). You will be asked to enter the street cross-section for each reach of the street (see *Typical Street Cross-sections* examples, this section). The *STREET FLOW* option allows you to add the runoff generated by the areas adjacent to the street. It also can be used to model a *V-GUTTER* street section which slopes towards the center of the street.

After determining that the street is filled to the maximum desired depth, you would select one of the other menu items to install street inlets (catch basins) and storm drain pipe, or a channel.

TYPICAL STREET CROSS SECT1



TIONS - HYDRAULICS



Addition of Runoff:

The ADDITION of RUNOFF option uses the current stream time of concentration for calculating rainfall intensity. The added input area and development type is then used to calculate the amount of runoff or added flow from a subarea. This option can be used after using either the ***PIPEFLOW, IMPROVED*** or ***IRREGULAR CHANNEL flow*** options to determine the time of concentration for the area flow being added.

Street Inlet + Parallel Pipe + Area:

The STREET INLET + PARALLEL PIPE + AREA option is similiar to the ***STREET FLOW*** option, except it assumes that a street inlet is to be installed at the top of this street segment or reach. This option uses the under street pipe flow travel time to determine the time of concentration used for rainfall intensity calculations. The following should be considered when using the ***STREET INLET*** option:

The longitudinal slope of the street for inlet calculations is determined from the elevations entered for the stations or point numbers. This slope determines the depth of flow in the street and through the area of the street inlet.

The capacity of the street inlet may be entered either manually or by using the D.O.T. HEC-12 manual calculations included in the program for curb inlets only. The program compares the street inlet capacity, and the capacity of the drain pipe(s) under the street. It then uses the lesser of the above capacities for the flow entering the street inlet, and assumes the remaining flow, if any, is continued in the street segment below the street inlet.

If the D.O.T HEC-12 curb inlet calculations option is used, the following should be noted. The program uses the street cross-section or cross slope data entered for normal street flow. However, you may modify this data by entering a Street ***Depression (see Typical Street Cross-sections*** examples, this section). The depression must be at least as wide as the gutter, but no wider than the distance from the curb to grade break. The depression depth is subtracted from the normal street gutter flow line adjacent to the curb, and added to the street cross section at the intersection of the width of the depression. The program then calculates the depth of flow through the depressed section and gutter to determine the curb inlet capacity.

You will enter the length of the curb inlet. The program first calculates the length required for total flow interception, then calculates the efficiency or amount of flow intercepted using the length of the inlets that you entered.

If the longitudinal slope of the street is **less than one percent,** the program considers this to be a sag location and calculates street inlet capacity using either the ***Weir*** or *Orifice* flow equations, considering the entered height and length of the curb opening.

Curb inlets may be installed on both sides of the street if the normal street flow was entered to flow on both sides.

The program will calculate the pipe size required to handle the street inlet flow rate, or you may manually enter the pipe size. The slope of the pipe is normally the same as the street;

however, you may override this value and enter a different pipe slope in percent.
If a confluence point is reached when using this option, the program will continue the pipe flow(s) below the confluence point. Therefore, the option may be used immediately after a confluence point and the sum of the preceeding pipe flows will continue under the street.

When designing a drainage system, the *STREET INLET* option provides a realistic method for design and evaluation of storm drain systems using streets, street inlets, and storm drain pipes. You may design the system using 10 year storm data, installing street inlets and pipes at points where the street flow exceeds the top-of-curb. Then, you can evaluate the same system using a 100 year storm, at AMC III, holding the pipe sizes to those used with the 10 year storm. The program allows you to freeze the pipe sizes when revising the *control data* or changing to a 100 year storm. It will then evaluate each street inlet, and limit pipe flow to a maximum pressure flow rate of that using the elevation difference as the head loss. The remaining flow will be left in the street. The results will show whether the depth of street flow exceeds the right-of-way limits.

Pipe Flow Travel Time:

The program calculates the size of pipes to the nearest 3 in. or 5 cm that will handle a nonpressure open channel flow using a *D/d* equal to 0.900. It will handle circular or elliptical shaped pipes. If the User *Input Size* option is used, the program first evaluates the pipe as an open channel. If the pipe is too small for nonpressure flow, it shifts to pressure flow calculations and and will calculate the aproximate hydraulic grade line required at the pipe entrance for pressure flow. Critical depth is calculated for open channel nonpressure pipe flow. The pipe flow option calculates the time of concentration from the velocity and distance of flow.

Improved Channel Travel Time:

The program calculates the depth of flow, velocity, and travel time through a trapezoidal, rectangular or V-shaped channel. You may also specify a box channel, and if the depth of flow exceeds the height of the channel the hydraulic grade line is calculated for the entrance of the channel. The critical depth is calculated for non-pressure flow conditions. Travel time and a new time of concentration is calculated.

Irregular Channel Travel Time:

Irregular channel shapes (up to 3 flow lines) are entered using the X-Y grid coordinates of the channel cross section. The procedures for entering irregular cross-section data are described on page 3 of Section 5. The travel time and a new time of concentration is calculated from average channel velocity and flow length.

Confluencing:

When reaching a point where two or more streams join, the *CONFLUENCE* option must be used. Here, you enter the total number of streams that are joining, and the individual number of the particular stream (number the streams starting at 1, in sequence up to a maximum of 5). Until the confluence is complete, you must start each added stream using either the *INI77AL AREA* or *USER INPUT* option and again route the added streams as appropriate down to the confluence point. After the last stream has been confluenced, you may continue routing the stream down to the next confluence point and adding subarea flow as necessary along with the routing process. When reaching the next confluence point, the sequence is started again.

> **Note: The MAIN STREAM** confluence option is normally not required. It should be used only when you reach a confluence point **and any of the additional incoming streams contain confluences upstream.** ***Additional incoming streams are*** defined as those that have **not** already been entered to the confluence point. If the MAIN STREAM confluence option is used, you must number the mainstreams in sequence, starting at 1 up to a maximum of 5. For additional explanation of ***Main Stream*** *Confluencing*, ***see Junctions*** paragraph in Chapter 6, Section 11.

Completing Each Menu Item:

As you build the data file, the results for each option selected are displayed in detail on the CRT. After finishing the option and reviewing the results, you may:

Accept the results, and the entered data will be stored in the data file.

Change any, or all of, the data before it is stored.

Select and use another option.

After all desired menu items have been run, you can complete the file by entering [RI or pressing the [ESC] key to return to the main menu.

Reports

There are three report options as show on the main menu.

Item 2 on the main menu provides a detailed report that contains the same data as appeared when building the file. This report may either be sent to a printer, or to an output file for later viewing or printing.

Item 3 on the menu provides a summary form report which requires a printer width of 132 characters. This report may also be saved as a file or sent to the printer.

Item 6 on the menu provides only a listing of the control data and options that were entered into the data file. Calculations are not included. This report is useful when editing the file.

Revising the Study Data File

You may make changes to, insert options, or delete options that were used in the program. When Menu Item #4 is selected from the main menu, the *control* ***data will*** be displayed and should be revised, if necessary. Then a listing of the options used will be displayed by line number. You can revise an option by entering its line number, add an option above or below the entered option line number, or delete the option. If you are revising an option, the previously entered data is displayed in reduced intensity, and if the data is O.K., just press RTN. If a change is required, enter the new data and the old information will be replaced.

Add to Study Data File

When this menu item is selected, the program will first run calculations for the existing data up to the last option entered, displaying the results on the CRT. This is necessary to update the previously entered data. Then the program shifts to the standard ***Build File*** mode, and you can then add new data. When the last new option has been entered, enter R or press the ESC key to return to the main menu.

Special Notes:

Detailed rational program output files require normal SO-character width paper. The summary form printouts require 132-character width paper (15” wide, OR COMPRESSED FONT).

(This page intentionally left blank)

Section 2

Unit Hydrology Programs

When [2] is entered from the Hydrology/Hydraulics Menu (see Overview), the CRT will display the following menu.

```
            CivilCADD/CivilDESIGN Engineering Software

        UNIT HYDROGRAPH HYDROLOGY PROGRAMS:

        The following programs are available:

        1 - Universal (Includes San Diego Co.).
        2 - San Bernardino County.
        3 - Riverside County.
        4 - Orange County.
        5 - None, Exit program.

                    Enter program option desired >
```

Select and enter the menu number that corresponds to your requirement.

DATA REQUIRED:

You must obtain rainfall isohyetal data, soil, and elevation data for the study area from county or U.S. NOAA documents before using these programs. (Note: The Orange County program has the rainfall data included, and some programs do not require elevation data). All the programs will accomplish *area averaging* of soil and rainfall data, and will calculate the soil loss rates for the program run based on this data. You may, however, over-ride the program soil loss figures.

PROGRAM METHODS:

LAG TIME: There are many methods of calculating the lag time (time from beginning of effective unit rainfall to that instant the summation hydrograph reaches 50% of ultimate discharge). Some agencies require that it be determined by a rational analysis of the basin for a time of concentration and then multiply the 'TC' by a constant to find the lag time. Other agencies require use of an empirical formula based on distances and elevations to obtain the lag time. The various unit hydrology programs contained in this package offer all of the available options.

S - GRAPHS: All of the unit hydrology programs use an S-Graph summation hydrograph approach where the watershed discharge, in percent of ultimate discharge as a function of time expressed in percent of lag, is stored in or calculated by the program. The stored S -

Graphs are those developed for Valley, Valley Developed, Foothill, Mountain, or Desert areas. The ***Soil Conservation Service (SCS)*** Dimensionless S - Graph is also stored in the in the Universal Unit Hydrograph program. Here you may also manually enter a unit hydrograph, and the program will convert this to an S-Graph for flood hydrograph calculations. Using the S - Graph, the program takes the unit interval entered, and calculates the time percent of lag time this unit interval covers, then integrates the S - Graph to determine the accumulated average percentage of ultimate discharge for each time period. The incremental average percentage of ultimate discharge for each unit period is calculated by a series of successive subtractions.

NOTE: Your selection of unit time interval should be in the range of 15 to 25% of the watershed lag time.

RAINFALL DISTRIBUTION: Depending on your county or regional requirements, the programs require rainfall data for either the total storm or for specific time periods in the study event year. The programs then adjust the rainfall value(s) for area1 effects. These area1 reduction amounts depend on your county, or SCS, or NOAA atlas requirements for the particular program being used. The programs then distribute the rainfall to the time periods required by your county or reviewing agency.

RUN-OFF **HYDROGRAPH** DEVELOPMENT: Various methods of soil loss rate or infiltration rates are applied to the rainfall amounts obtained above. The programs compute the effective rainfall for each time period, and then multiplies this value times the unit hydrograph figures to determine the flood hydrograph which would result for that rainfall increment. This process is repeated for each succeeding rainfall value, and the flow ordinates are summed to find the total flow for each unit time period.

PROGRAM OUTPUT FILES: The unit hydrograph programs produce an output file showing the input data, intermediate result data such as the unit hydrograph, and the final flood hydrograph with flow and volume data. The programs also produce two other output files, one with the job name and extension '.RTE' which is for use with the routing program. The other, with the extension '.HCU', can be used with the HEC1 program.

Section 3

Hydrograph Routing Program

This program assists in designing channels or structures to contain or attenuate flood hydrograph(s). It contains a Universal Flood Unit Hydrograph Routing program that assists in designing channels or structures to contain or attenuate flood hydrograph(s). In additional to instructions in this section of the manual, the program has a help file which may be viewed or sent to your printer by selecting option [0] when entering the ***Universal Routing*** program.

For Southern California users, this level also includes a program that models retarding basins using an ***F0601 Hydrograph file*** as the input hydrograph. Documentation for this program is found in file *RETARD.DOC*, which can be printed if desired.

Enter [3] from the Hydrology / Hydraulics Menu (see Overview) to access this level.

DATA REQUIRED:

The routing program uses flood hydrographs, either manually entered or created from the Unit Hydrograph programs. The hydrographs must be created, or entered, with a constant time interval. This interval may be as small as 1 minute, or as large as 60 minutes. The program does allow mixing of hydrographs with different time intervals. The maximum size flood hydrograph is 1000 time intervals.

PROGRAM METHODS:

FLOOD HYDROGRAPHS: To run the program, you must first have an initial input hydrograph. This is usually obtained automatically if one of the Unit Hydrology programs are run. Or, you may manually enter a hydrograph into the program. You can also combine or add hydrographs to the initial hydrograph.

STREAMFLOW ROUTING: The stream routing option allows you to use the following types of channel flows after being given the length of the stream reach and elevation difference:

> Open Channel: For open channel flow, you can use the Irregular Shape (see explanation under Rational Programs), Rectangular, Trapezoidal, or 'V' shaped channels. The program will advise you of the flow capacity if the channel capacity is exceeded. You must adjust or increase the size of the channel to allow the flow of average over 50% peak value for the program to compute stream flow data.
>
> Pipe or Box (Non-pressure, or with Pressure & Storage): Here, the program will first attempt non-pressure flow through the pipe or box, then notify you if pressure flow is necessary and give you the option of using a larger size or to use pressure flow. If pressure flow is used, the program will compute the Storage requirements at the Headworks for the hydrograph being used.

After the channel data has been entered, the program computes the channel outflow based on the Soil Conservation Service (SCS) Convex routing method. If the travel time through the channel exceeds the time interval of the input hydrograph, you have the option of delaying the hydrograph outflow by the given number of hydrograph increments.

RETARDING BASINS: The retarding basin option uses the Storage Indication Method of routing a hydrograph through a reservoir. This is also called the Modified Puls Method. To use this option, you must first determine the storage capacity of the reservoir (in Acre-Feet) versus the depth of water in the reservoir. Then, you have 2 options for entering or computing the amount of outflow of the basin versus depth. These options are:

> One: User entry of outflow values.
>
> Two: Computation of discharge for various outlets at different depths, including either weir (V or rect) flow; pressure/non-pressure pipe flow; free outlet (weir) pipe flow; or pressure/non-pressure box channel flow; depending on reservoir depth.

Option two allows you to enter up to 10 different type outflow devices at different heights above the basin bottom.

After the storage, depth, and outflow data are determined, the program prints the resulting outflow, and depth in the basin, at each time increment.

Note: The program will store water in a retarding basin if there is no outflow at depths greater than zero. Or, if the initial basin depth is set at greater than zero before the start of the inflow hydrograph, the program will add flow to the outflow hydrograph until the calculated outflow capacity reaches zero.

FLOWBY BASINS or SPLIT **FLOW:** The flowby basin is a structure designed to receive all flow in excess of a given peak rate from a flood hydrograph. The flowby basin is usually adjacent to a trapezoidal channel, which is designed to be small enough in size so that at the given peak flow rate the channel will overflow into the flowby basin.

This program allows you to either input the size of the trapezoidal channel and let the resulting peak flow rate be computed by the program, or you can manually enter the peak flow rate.

After the peak flow rate is entered, the program will store the overflow hydrograph from the flowby structure into another stream (number 1 to 5), and print out the resulting *main stream* hydrograph. You have the option of later recovering the excess flowby at its original rate, or at a constant rate entered by you.

After the peak flow rate is entered, the program will store the overflow hydrograph from the flowby structure into another stream (number 1 to 5), and print out the resulting main stream hydrograph. You have the option of later recovering the excess flowby at its original rate, or at a constant rate entered by you.

The split flow procedure allows you the option of diverting a percentage fraction of the main stream hydrograph into another stream (number 1 to 5). This fraction will be stored in the stream number until you recover the stream by using the combine option.

STORING HYDROGRAPHS: You have the option, at any time, to store the current hydrograph in a stream (number 1 to 5). When this is done, all previous data in that stream number is over-written by the newly stored stream, and the current flood hydrograph is erased. After using this option, you must use the Combine Option to put a new hydrograph into the stream before the stream routing, retarding, or flowby options may be used. This option may be used to temporarily set aside the main stream after a flowby basin, then use the combine option to recall the excess flow hydrograph from the flowby basin, then route this flow through a channel, then again combine the stored main stream with the current hydrograph to simulate the flood hydrograph with such a flowby structure.

COMBINING HYDROGRAPHS: This option permits you to add hydrographs from Unit Hydrograph program files, or add a manual hydrograph to the current main stream. Also, a stored hydrograph may be added directly (by a user defined time delay), or added at a constant flow rate as defined by you. When adding hydrographs, the program converts either the present mainstream hydrograph or the added hydrograph to the smallest time increment of the two.

VIEWING/PRINTING of **HYDROGRAPHS: The** program allows you to view the resulting hydrograph after each routing option. These results show the time, flow, and volume time increment of the flood hydrograph.

(This page intentionally left blank)

Section 4

L.A. County Water Surface Pressure Gradient

This level is used to describe WSPG, WSPGN and WSPGW water surface pressure gradient (WSPG) programs. Refer to the ".DOC" file for specific instructions of the program version you have.County, *CA.* **The use of these programs by others is made and accepted with the understanding that the Los Angeles County Flood Control District, and Joseph E. Bonadiman and Associates, Inc., makes no warranties, express or implied, concerning the accuracy, completeness, reliability, usability, or suitability, and the District and Joseph E. Bonadiman and Associates, Inc. shall be under no liability for any use made thereof.**

The programs compute uniform and nonuniform steady flow water surface profiles and pressure gradients in open channels or closed conduits with irregular or regular sections. The flow may alternate between super critical, sub critical or pressure flow in any sequence. Different types of channel cross-sections can be included, as well as bridge piers, entrances and exits; junction structures; transitions structures; multiple inflows; sudden contraction and expansion; etc. Critical depths, hydraulic jumps, etc. are also calculated.

A module has been added which allows you to input data directly without having to *format* ***cards,*** and to also assist in editing the input data. A Help? routine is also included in this input/edit module.

NOTE: L.A.C. program printouts require **132 character** width paper (Compressed Font).

Water Surface Pressure Gradient Programs

Basic Theory: The computational procedure is based on solving ***Bernoulli's Equation*** for the total energy at each section, and ***Manning's Formula*** for friction loss between the sections in a ***Reach. The*** open channel flow procedure uses the standard Step Method. *Confluences* and bridge piers are analyzed using pressure and momentum theory. Program uses basic mathematical and hydraulic principles to calculate data such as cross sectional area, wetted perimeter, normal depth, critical depth, pressure, and momentum.

Computational Procedure

Input preparation: The channel or conduit system is initially subdivided into following elements: ***System Outlet, Reach, Transition,*** *Confluence* (junction), ***Bridge Exit,*** *Bridge En-* ***trance, Wall Entrance*** (sudden contraction), ***Wall Exit*** (sudden expansion), and ***System Headworks.*** Each element is internally assigned a number. The input data must consist of a minimum of three elements ***(System Outlet, System Headworks,*** and any other element) and is limited to a maximum of 200 elements. If more than 200 elements are needed, the overall system must be broken into two or more systems. The station and invert elevation of all channel points starting from the ***System Outlet*** **up to the** ***System Headworks*** are required. Also, the channel type at each element point is required. ***System Outlet*** must be the first element entry. Each element must include one of the Data Element Options shown below.

Data Element Options:

Reach	Transition (Gradual Change)
Bridge Exit	Bridge Entrance
Wall Exit (Sudden Expansion)	Wall Entrance (Sudden Contraction)
Confluence (Junction with up to 2 laterals)	

The final element entry is the ***System Headworks.***

Flow Rates: The starting flow rate (Q) at the upstream terminus of a system is specified with a Q ***Card*** entry. The flow rate (Q) is increased at the desired locations by specifying lateral inflow rates using ***JX Card*** entries. The flow rate can be reduced by using a negative lateral flow (Q). This reduction, or loss, is intended to account for channel storage. If it is used in cases where the channel or conduit branches, it should be understood that no loss will be computed.

Multiple Profiles: To obtain additional water surfaces or pressure gradient profiles for different flow rates in the system, a maximum of five flow-rate Q ***Card*** entries may be included.

Manning's 'n': The program uses the ***Manning Formula*** for the friction loss in all types of conduits or natural channels. The program can only take one 'n' value per element; however, the 'n' value can change for subsequent elements. If a section has lining composed of different roughness coefficients, a composite 'n', based on anticipated depth of flow, should be hand computed. If an 'n' value is not specified with the input data, the program uses a value of 0.014.

Water Surface Controls: Water surface controls at the downstream terminus (System Outlet) or the upstream terminus (System Headworks) are optional input values. If water surface controls are not entered, the program will use critical depth controls.

Critical and Normal Depths: Critical depth is computed for every section for the given flow (Q) using the ***Specific Energy Equation.*** Normal depth is computed in every ***Reach*** element on a positive slope for the specified flow (Q).

The velocity head (Hv) is computed using the mean velocity of the section. This may not be accurate in the case of a complex section, such as one with shallow flow in the horizontal overbank area where velocity distribution is not uniform. If the program is to be used in this situation, you should be aware that some error may be introduced in the results. A check on the magnitude of the error can be made by using the parabolic method to determine specific energy.

Water surface Stages: The lower stage water surface profile begins at the ***System Headworks*** and ends at the ***System Outlet. The*** computation will proceed downstream in every consecutive element as long as energy is available to maintain flow in the supercritical stage. When energy becomes expended at any point in an element, the lower stage profile will be discontinued from that point to the downstream end of that element. Then computation will resume in the element with a critical depth control until the ***System Outlet*** is analyzed.

The upper stage water surface profile begins at the ***System Outlet,*** and ends at the ***Headworks.***

Computation proceeds upstream in every element as long as the water surface at the downstream end of any two adjacent elements can support the moving mass of water to flow at the critical or subcritical depth. Otherwise, computation will be discontinued from the downstream point to the upstream end of that element. Then computation will resume at the downstream end of the next element with critical depth control, provided no depth less than critical depth has been computed at that point on the lower stage profile. Then computation will proceed upstream until the ***System Headworks*** is analyzed. Note that if the computed depth of flow in any open section exceeds the given section height, the program will assume an additional lo-feet of vertical wall. The exception to this is Channel Type 1 (see Figure 6-1), where the side slopes are extended outward until the additional lo-foot vertical height is reached.

The jump routine begins at the ***System Outlet*** and ends at the ***Headworks.*** It searches the lower stage and the upper stage profiles for points of equal energy. If a jump is encountered, it will be approximately located, and data on either the upper stage or lower stage not consistent with the greater energy theory will be deleted from every element. The final profile will be a composite of upper stage and lower stage with hydraulic jumps in between.

Data Processing System Description

File Structure: The WSPG program uses a formatted input data file. The file should be built and/or edited using the ***WSPG.EXE*** program included in this package. You may construct the file using ***VUE*** or a ***Word Processor,*** if **you** are thoroughly familiar with the ***formatted*** file structure; however, if you do so, and if the data elements are not precisely located in the correct ***card columns,*** the program will not run properly.

The term ***Card*** is used often in describing this program. The original WSPG programs ran on a main frame computer, with ***punch card*** input rather than CRT input.

Input/Output File Names: You will be asked for a study name of up to **6** characters. The input data file will be this name plus a ***.WSP*** extension. The output data files will have a *.EDT* extension for the edit program output and a *.OUT* extension for the final output file.

Data Entry: Required input into the system consists of:

Title Information.
Channel Element Definitions.
Cross-section Definition.
Cross-section Points Definition.
Flow (Q ***Card),*** of which the first is required and the subsequent flows **(Q *Curds)*** are optional for change of flow rate in the system.

You may enter up to 3 lines of title information or leave the title lines blank. Then the program asks for ***System Outlet*** station, elevation, and channel description data. Following this, the program will display the list of channel element options as shown in ***Data Element Options:*** above. The general data required for each Element will be the Upstream station in feet, invert elevation in feet, and the channel description. You should use the ***default*** entry of the last channel description entered if no change in size or cover or piers are made. When a channel description (channel size), for example a 3 foot pipe, has been entered, the channel is assigned an identification number. This same number may then be used repeatedly if the same size channel is encountered later in the system.

When reaching the *System Headworks, you* should enter the last station, invert elevation and channel description (these are also the *default* entries). Following this, the flow rate data at the *System Headworks* is required. Up to five flows may be entered, which if used, will result in 5 separate program runs. The flow rates entering at junctions are initially entered when entering data for a junction element. If more than one flow rate is entered at the *System Headworks,* the program will ask for a factor to modify the junction flow rates. If a factor of 1.0 is used, no change in junction flow will be made.

Editing Input Data File: After you have created the input data file, the *EDIT* program (provided with the *WSPG* programs, must be run to detect any errors and prepare the data files for the main program run. The entire input is thoroughly scanned for required information and range of values for optional information before processing begins. If any errors are found, processing will stop. Warnings may be issued, but they will not prevent processing. Error messages and Warning messages are found at the end of this section.

Running WSPG: After the edit program program has been run and ALL errors corrected, you may run the program with output directly to the screen, or to a an output file, or directly to the printer. This is accomplished by using the *COMP* program, which is included in your *WSPG* package. To printout the results file with correct page and line feeds, the printer must be set for 132-characters wide. If the *send to default printer* option is used, the form or page feeds are automatic. If an output data file is made, the form feeds and line feeds are printed as “O’s” or "1’s" as the first line character in the output file. These previously run files may be correctly printed or formatted by selecting the appropriate option when entering the program.

Processing consists of three phases, which are analysis of the system in the downstream direction (Phase I), analysis of the system in the upstream direction (Phase II), and analysis of the downstream profile (from Phase I) and the upstream profile (from Phase II) to obtain a composite profile (Phase III). The processing is designed to continue calculating unless gross errors are encountered. Warning messages may be issued concerning tolerance levels not being reached on an iterative approximation. These may, or may not, affect the overall solution to the problem; however, processing will continue. If gross errors are encountered, an error message will be issued and processing will stop.

Output of the system consists of three reports:

A listing of input, with edit scanning messages.

A water surface profile listing of the composite profile obtained in Phase III of processing.

A profile plot of water surface and channel geometry.

Program Limitations

A maximum of 200 elements are allowed.

A maximum of 50 intermediate points can be computed in a ***Reach*** element.

Critical depth cannot exceed 100 feet.

Program will not compute the water surface profile when the friction slope is at one or greater.

Open channel processing is limited to a depth of ten feet above the height of the described element.

Undulating bottoms cannot be calculated properly in an irregular shaped section, unless the depth of flow is above the undulations.

The program will not accept vertical drops in invert elevations.

Calculations in the water surface profile report may be slightly inaccurate (to O.OOl), due to rounding variables to be contained in the upstream and downstream data files.

The invert cross-fall 'e', in inches, is optional, and may be used only with Channel Type **#2**, a rectangular open channel or R.C. Box Sections.

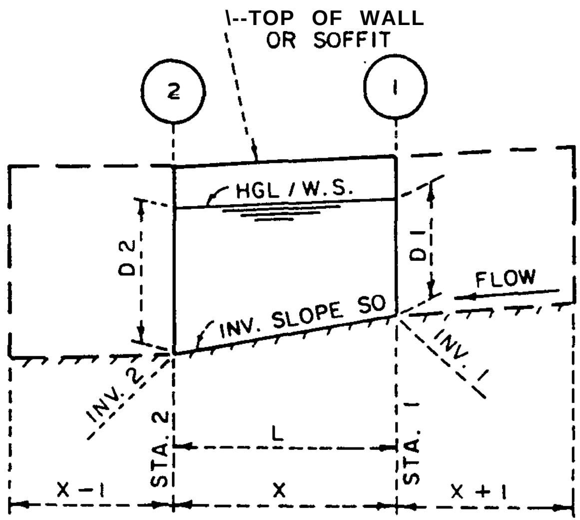
Element Description

The channel, conduit, or natural river system is subdivided into elements, as stated in the ***Input Preparation*** paragraph above. The program internally numbers the elements beginning with the ***System Outlet*** as number 1. Each successive element is numbered continuing upstream to the ***System Headworks. The*** maximum number of elements is 200.

Boundary Lines (see Figure 5-l): All elements are bounded on the upstream end by Section 1 and the downstream end by Section 2, except ***System Outlet*** and ***System Headworks*** which only have Section 1. You will enter data such as base width, conduit height, etc. for Section 1 of every element. The data for Section 2 for every element is taken by the program from the upstream Section 1 of the adjacent downstream element. Elements may have considerable

Continued on next page

ELEMENT BOUNDARY LINES



ELEVATION

FIG. 5 - 1

length between Section 1 and Section 2, as in a *Reach* element, or may have a zero length, as in a *Bridge Entrance* element.

L = Length of Element.
X = Number of the Element under Consideration.
X+1 = Adjacent Upstream Element.
X-l = Adjacent Downstream Element.

System Outlet (SO) (see Figure 5-2): The *System Outlet* is the downstream terminus of a channel. X is equal to one. X+ 1 can be any element except a *System Headworks.* Note that the element length is zero.

System Headworks (SH) (see Figure 5-3): The *System Headworks* is the upstream terminus of a channel. Element X-l can be any element except a *System Outlet.* Note that the element length is zero.

Reach (R) (see Figure 5-4): The *Reach* element is the length of channel, drain, or natural river with a constant invert slope, flow (Q), cross-section, and *Manning's 'n'.* A *Reach* may have a straight or curving horizontal alignment; however, a curved *Reach* must coincide with the beginning and end of the curve. The same applies to an angle point in the horizontal alignment-- a *Reach* must end or begin at the angle point.

In open channels (regular rectangular or trapezoidal sections), the superelevation of the water surface is computed and printed for each point in the curve. In pressure flow, bend losses, angle point losses, and manhole losses are computed and added to the friction loss for the *Reach.*

Element X+ 1 can be any element except a *System Outlet.*
Element X-l can be any element except a *System Headworks.*

Junction Structure (JX) (see Figure 5-5): The *Junction Structure* element is used where there is lateral inflow into the system. Two different laterals can be handled by this element. Element X-l can be any other element except a *System Headworks.* Element X+ 1 may be any other element except a *System Outlet.*

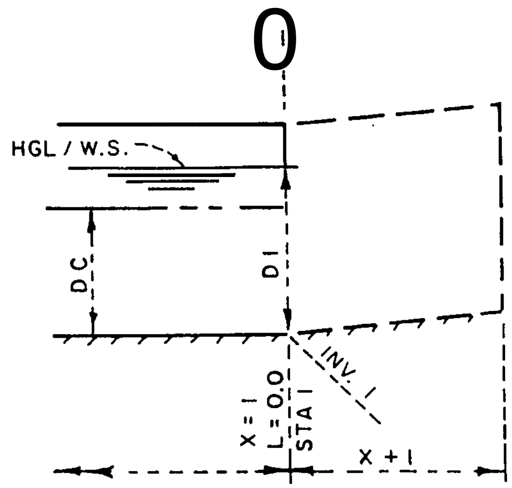
Transition Structure (TS) (see Figure 5-6): A *Transition Structure* is a gradual expansion or contraction from Section 1 to Section 2. The length (L) may be any positive number. Element X+ 1 may be any element except a *System Outlet.* Element X-l may be any element except a *System Headworks.*

Bridge Entrance (BE) (see Figure 5-7): A *Bridge Entrance* is an element used where flow enters from an element without piers into an element with piers. A *Bridge Entrance* is considered to be a zero length element, even though the bridge pier nose may have a minor length.

Element X-l may be a SO, R, JX, or TS.
Element X+ 1 may be a R, JX, TS, or SH.

Note that neither Section 1 nor Section *2* can be a pipe.

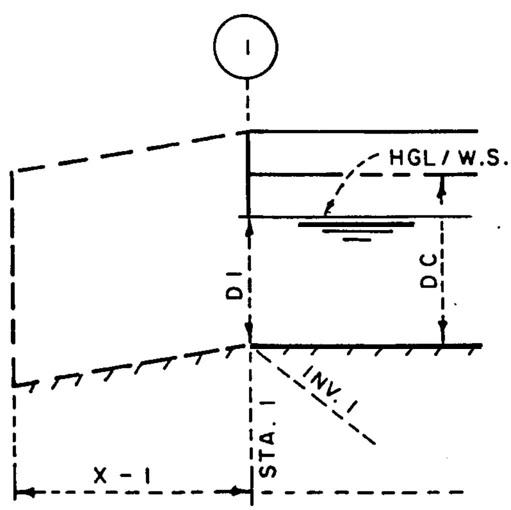
SYSTEM OUTLET



ELEVATION

FIG. 5-2

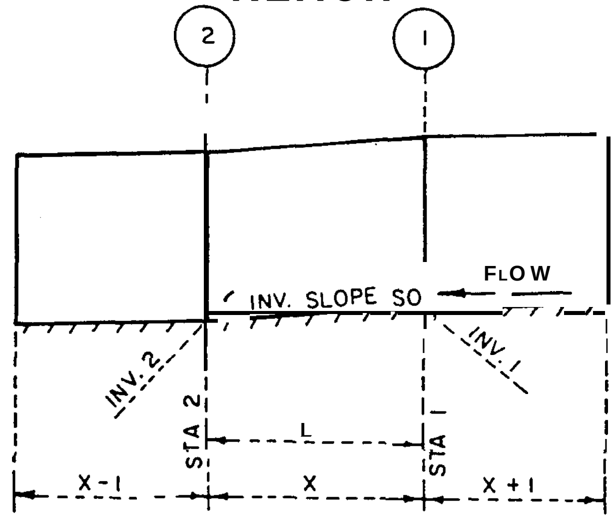
SYSTEM HEADWORKS



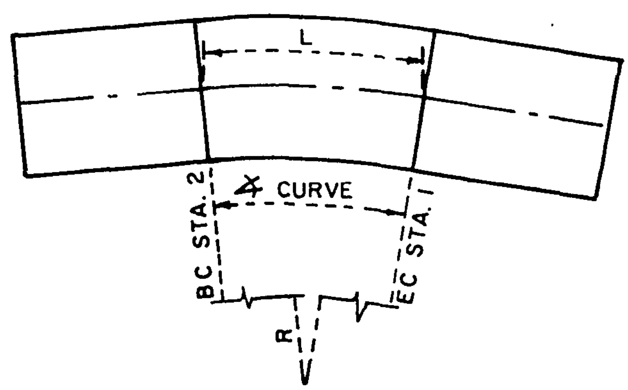
ELEVATION

FIG. 5-3

REACH

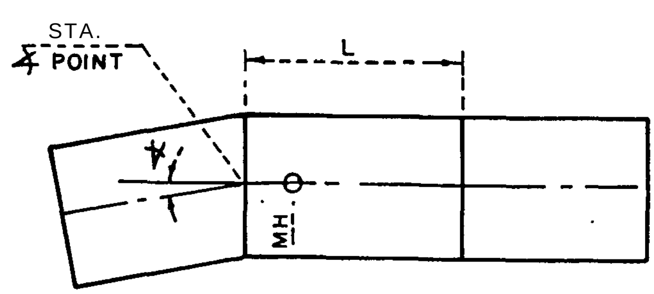


ELEVATION



PLAN A

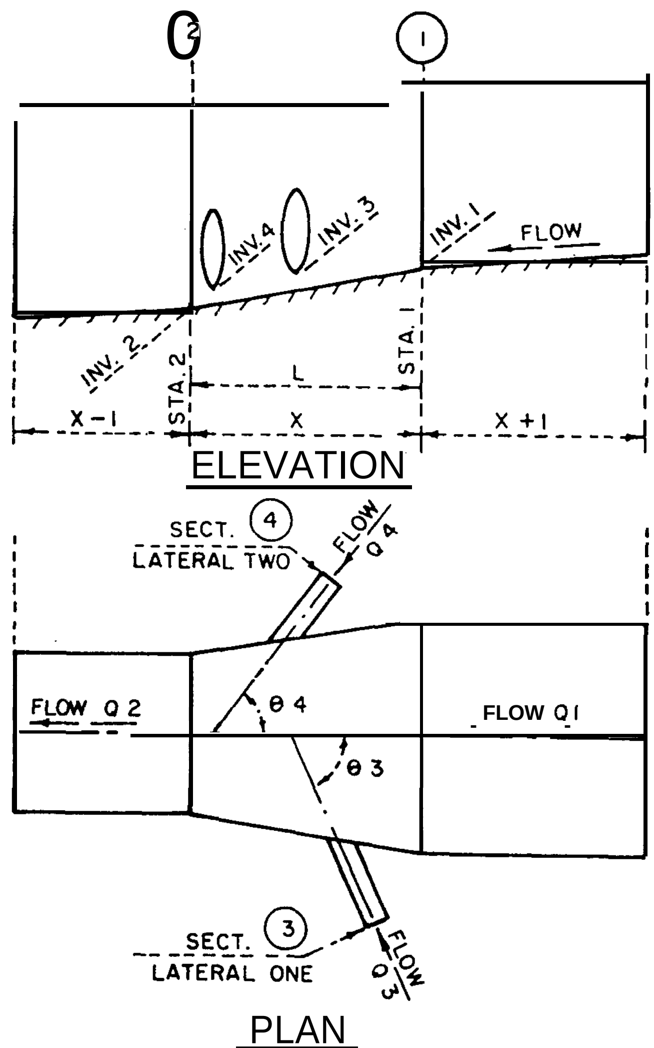
(EXAMPLE OF CURVED ALIGNMENT)



PLAN B

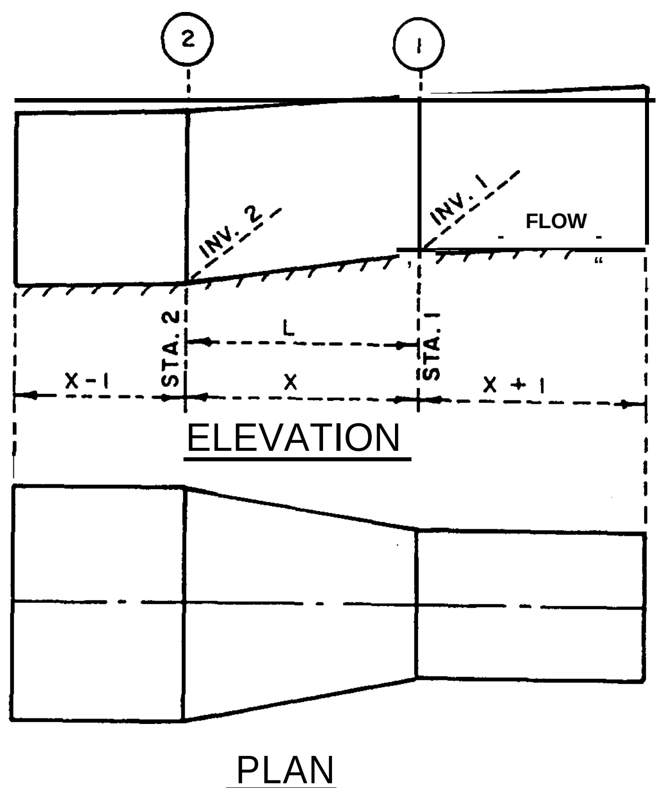
(EXAMPLE OF A STRAIGHT REACH
WITH ∡ PT. AT D/S END)

FIG. 5 - 4



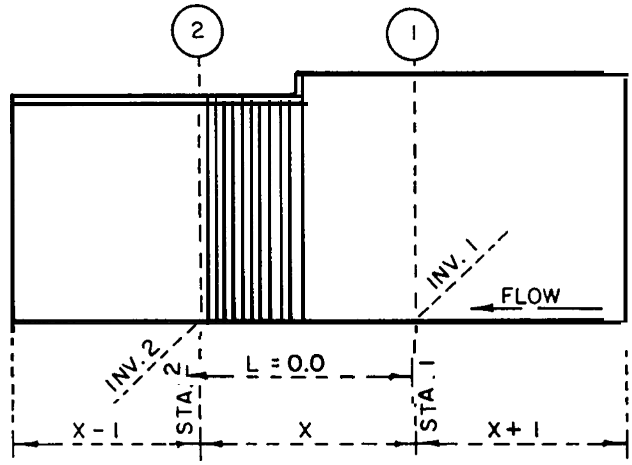
JUNCTION

FIG. 5-5

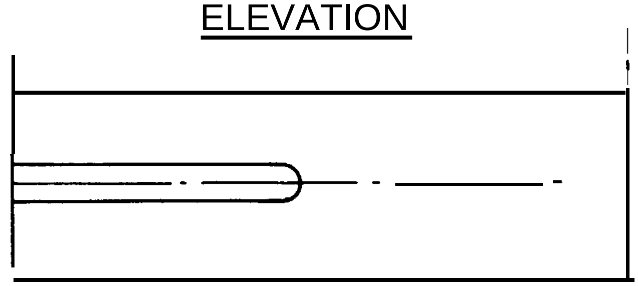


TRANSITION

FIG. 5 -6



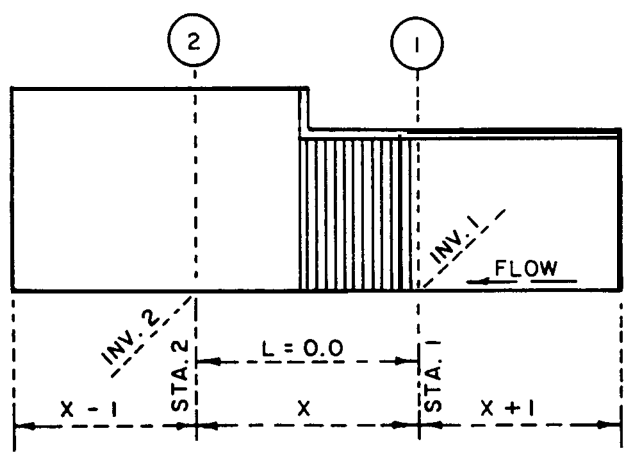
ELEVATION



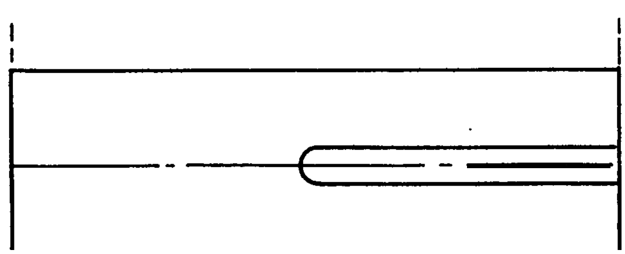
PLAN

BRIDGE ENTRANCE

FIG. 5 -7



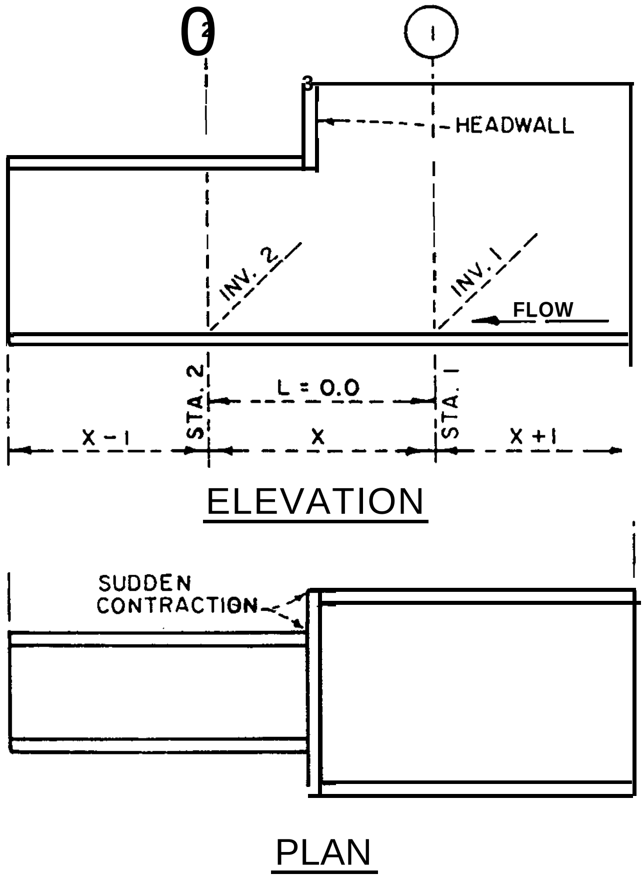
ELEVATION



PLAN

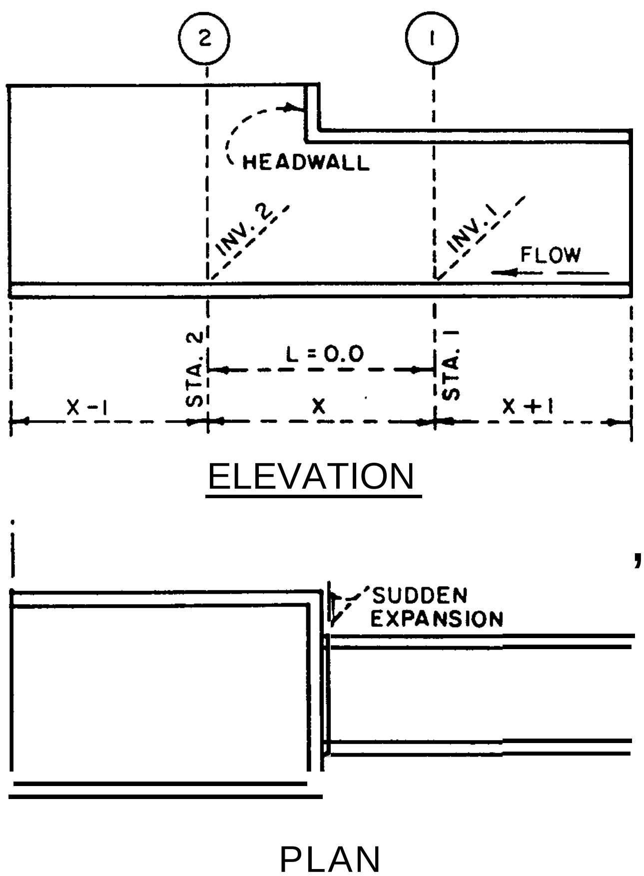
BRIDGE EXIT

FIG. 5 - 8



WALL ENTRANCE ·
SUDDEN CONTRACTION

FIG. 5 - 9



WALL EXIT .
, SUDDEN EXPANSION

FIG. 5 - 10

Bridge Exit (BX) **(see** Figure 5-S): The *Bridge Exit* is also **considered to be a zero length element. It** is used where flow exits from an element with piers into an element without piers.

Element X-l may be a SO, R, JX, or TS.
Element X+ 1 may be a R, JX, TS, or SH.

Note that neither Section 1 nor Section 2 can be a pipe.

Wall Entrance (WE) (see Figure 5-9): The *Wall Entrance* element is used when there is a sudden change in the conduit section, such as a headwall or an abrupt contraction. This element is a zero length element. You should supply the loss coefficient 'Kc' expressed in terms of the velocity head. If you do not enter a 'Kc' value, the program will use 0.5 for 'Kc'.

Element X-l may be a SO, R, JX, or TS.
Element X+ 1 may be a R, JX, TS, or SH.

The section for Element X+ 1 camlot have piers; however, it can be an open channel or closed conduit. The section for Element X-l can also be an open chamlel or closed conduit, and it can be with or without piers.

Wall Exit (WX) (see Figure 5-10): This element is used when there is a sudden expansion from a smaller to a larger channel or conduit section. It also has a zero length.

Element X-l may be a SO, R, JX, or TS.
Element X+ 1 may be a R, JX, TS, or SH.

The section for Element X+ 1 can be an open channel or closed conduit, with or without piers. The section for Element X-l can also be an open channel or closed conduit; however, it cannot have piers.

Channel and Conduit Section Description

Channels and conduits sections are classified as regular or irregular sections. The regular sections (Channel Types #l-#4) are trapezoidal and rectangular channels, box conduits, and pipes. The irregular sections (Channel Types #5 and #6) can be natural river sections or irregular shaped improved sections, with or without a cover. Piers or center walls can be included in any section except a pipe section.

Regular Channel Type Sections: The program uses the following regular sections:

Type#l Trapezodial open top, with or without piers. See Figure 6-l.

Type #2 - Rectangular open top, with or without piers. See Figure 6-2.

Type #3 - Box, covered trapezodial or rectangular, with or without piers. See Figure 6-3.

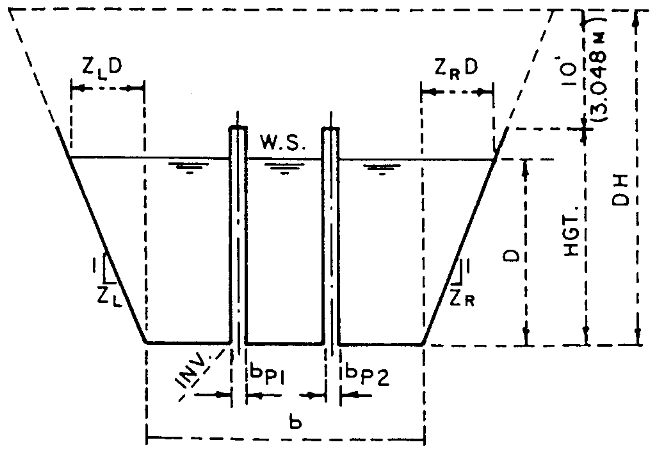
Type #4 - Circular pipe, on cell only. See Figure 6-4.

REGULAR CHANNELS

CHANNEL
TYPE 1

Trapezoidal section with or without piers.

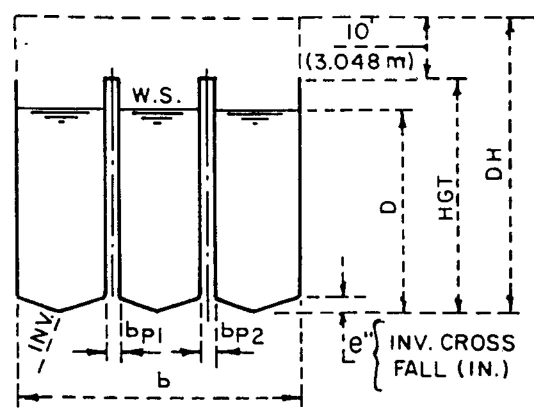
FIG. 6-1



CHANNEL
TYPE 2

Rectangular section with or without piers.

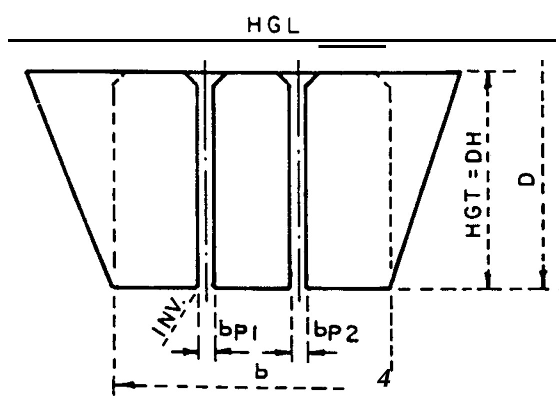
FIG. 6-2



CHANNEL
TYPE 3

Box **culvert, covered trap or rectangular section with or without piers.**

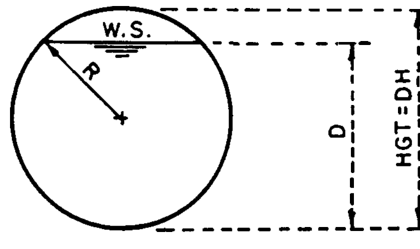
FIG. 6-3



CHANNEL
TYPE 4

Circular section (pipe) one cell only.

FIG. 6-4



Note: In multiple cell sections the cells may have variable width, but must be of equal height and have the same invert elevation. The top elevation of all piers in both regular or irregular channels are assumed equal.

Irregular Channel Type Sections: The program uses the following irregular cross-sections:

Type #5- Irregular open top, with or without piers. See Figure 6-5.

Type #6- Irregular covered top, with or without piers. See Figure 6-6.

Definition for Irregular Sections: An irregular cross-section (facing upstream) is defined by 'x' and 'y' coordinates of points P (x,y), in a counterclockwise direction from point P = 1 to point P = z (minimum of 3 points and maximum of 99 points).

Point P = 1 (x,y) is where
x (1) = x min and if x(2) is also x minimum then
y (1) is greater than y (2)

Limitations for Irregular Sections:

Location of x and y axis: The center of the reference axis (x=0,y=0) must not fall on the perimeter of the cross-section.

Flow Line: A section can have only one low flow channel.

Section Shape: A section is allowed one minimum and maximum in the x and y directions. For example, between points from x minimum to x maximum, the consecutive values of x must be equal or greater. From x maximum to x minimum, the consecutive values of x must be equal or smaller. The same holds in the y direction.

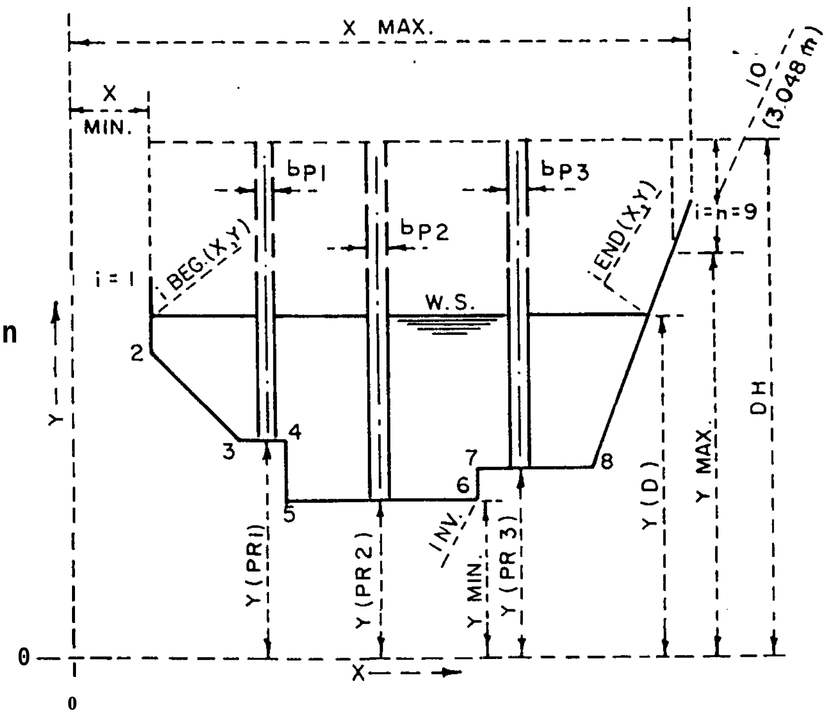
Piers: The reference (x,y) axis for piers must be the same as used for the cross-section. The y values are entered from left to right.

IRREGULAR CHANNELS

CHANNEL
TYPE 5

Irregular open top section with or without piers.

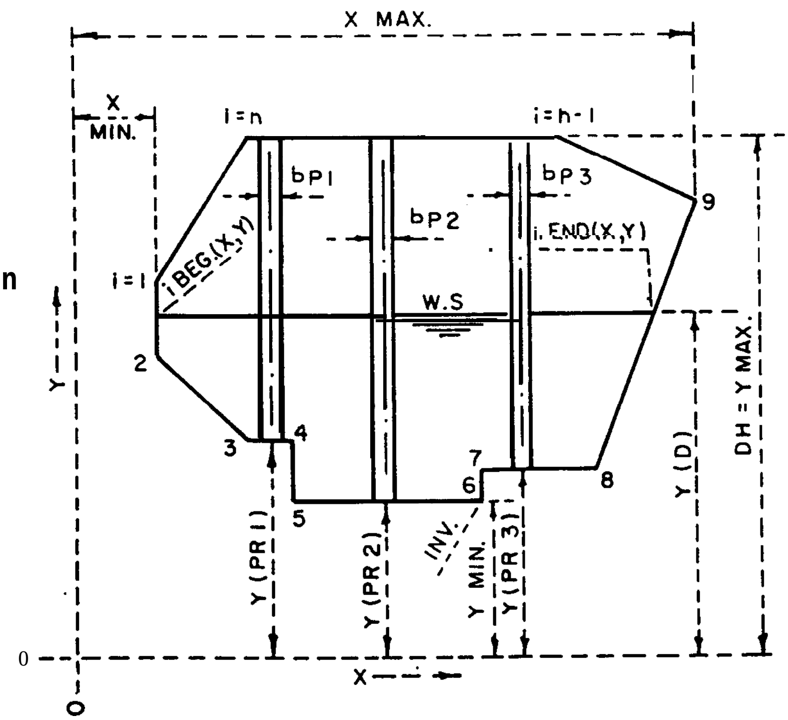
FIG. 6-5



CHANNEL
TYPE 6

Irregular covered section with or without piers.

FIG. 6-6



Computation Procedures

Symbol Defmitions:

Symbol	Definition
A	Cross sectional area of flow
AP	Cross sectional area of pier
b	Base width of channel
bnet	Net base width of channel
Ca	Central angle of bend in degrees
D	Depth of flow
Dc	Critical depth
DH	Maximum open flow depth in a section: Ten feet above channel height in open section, or Height of channel in closed section
df	Deflection angle in degrees
DN	Normal depth
E	Specific gravity
EGL	Energy Grade Line elevation
e	Invert cross fall in inches. May be used with channel type two only (optional)
F	Force
g	Gravitational constant (32.2 ft/sec/sec)
H	Drop in invert between two points
HAPT	Head loss due to angle point
HB	Head loss due to bend or curve
HF	Head loss due to friction
HJ	Head loss in a junction
Hm	Miscellaneous losses
HMH	Head loss in a manhole
Ht	Head loss in a transition
H-V	Velocity head
INV	Invert elevation in a channel section
Kc	Minor loss factor at wall entrance
L	Length between two stations
M	Momentum
n	Manning's 'n', coefficient of roughness
P	Hydrostatic pressure
Q	Flow rate
R	Radius of pipe
RI-I	Hydraulic radius
r	Radius of curve on horizontal alignment
Sfav	Average friction slope between elements
SF	Friction slope (energy loss per foot)
SC	Critical slope (slope at critical depth)
so	Invert slope
SE	Super elevation
STA	Station

V	Velocity
WP	Wetted perimeter
WS/HGL	Water surface or hydraulic grade line elevation
ZL	Left side slope
ZR	Right side slope

The above symbols may have one of two suffixes:

1: Identifies the variable at the upstream end of an element.
2: Identifies the variable at the downstream end of an element.

Example: V1 = velocity at the upstream end, and V2 = velocity at the downstream end.

Assumptions: Steady one dimentional flow and incompressible fluids.

Basic Equations of Steady Flow:

Equation of Continuity:

$A1\ V1 = A2\ V2 = Q$

Manning's Formula (friction slope):

$Sf = \{Q\ n\ / [1.486\ A\ (RH)^{2/3}]\}^2$

Bernoulli's Equation (open flow):

$D2 + HV2 + \Delta L\ Sfav = D1 + HV1 + \Delta L\ So$ where $HV = V^2 / 2g$

Bernoulli's Equation (pressure flow):

$D2 + HV2 + \Delta L\ Sfav + Hm = D1 + HV1 + \Delta L\ So$

Angle Point Loss:

$Hapt = 0.0033\ df\ HV$ (Note: District recommends df not to exceed 6 degrees.)

Bend Loss:

$HB = 0.2\ HV \sqrt{Ca / 90}$

Manhole Loss:

$Hmh = (0.05\ HV)(No.\ MH)$ where No. MH is number of manholes in a reach.

Specific Energy:

$E = D + HV$

Pressure - Momentum:

$P2 + M2 = P1 + M1 = F$ where $M = (Q)^2 / (Ag)$

Critical Depth:

D_C is the depth of flow at minimum energy. To find D_C by parbolic method, see *Critical Water Surface by Minimum Specific Energy Using the Parabolic Method,* Hydrologic Engineering Center, U. S. Army Corps of Engineers, Eichert, Bill S.; or otherwise iterate for D_C in the *Specific Energy* equation.

$E_C = F(D_C) = D_C + HV_C$

Normal Depth:

Dn is the depth of uniform flow and is found by iteration from *Manning's* formula.

$A\,(RH)^{2/3} = F\,(Dn) = [Q\ n] / [1.486\, So^{1/2}]$

Reach Analysis

Open Flow:

Intermediate points are computed on the Water Surface profile in a reach using the standard step method. The difference in velocity head between two adjacent points is held to a maximum of ten percent.

$\Delta L = (E2 - E1) / (So - Sfav)$

Pressure Flow:

EGL1 = EGL2 + HF + Hm
D1 = EGL1 - HV1 - INV1

If Water Surface profile rises to the soffit of a conduit before the end of the reach, or if the HGL breaks seal before the end of the reach, minor losses are adjusted to reflect only the portion of the reach under pressure.

Super Elevation

Super Elevation (S.E.) is computed in curving channel as follows:

Channel Type 1 (Trap. Section):

Subcritical flow: S.E. = 1.15 [HV / r] [b + D (ZL + ZR)]
Supercritical flow: S.E. = 2.6 [HV / r] [b + D (ZL + ZR)]

Channel Type 2 (Rect. Section):

Subcritical flow: S.E. = HV b/r
Supercritical flow: S.E. = 2 HV b/r

Transition Analysis

If V2 is greater than V1 then Ht = 0.1 [HV2 HV1].
Otherwise, Ht = 0.2 [HV1-HV2]

Junction Analysis

ΔY = [(Q2 V2) - (Q1 V1) - (Q3 V3 COS03) (1/g) (l/A ave) +ΔL Sfav
where A ave = [(Al + A2)/2], and
ΔY = D1 +ΔH - D2

HJ = Y + HV1 + I-IV2

Wall Entrance Analysis (Sudden Contraction)

Lower Stage Profile (Upstream (U/S) Control):

Find depth at the Downstream (D/S) end by iteration in the following equation.

M2 + P2 = Ml [(Al - A1Wall)/A1] + P1 - PlWall
where Al wall is the area of the obstructed part of Al, and PlWall is the pressure on the obstructed part of Al.

Upper Stage Profile (Downstream Control):

If the control depth is less than the conduit height, find the depth at the U/S end from:

M2 + P2 = Ml [(Al - A1Wall)/A1] + P1 - PlWall

Otherwise, find D1 by iteration from the following equation:

D2 + HV2 + Kc ABS [HV2 - HV1] = D1 + HV1
where Kc ABS [HV2 - HV1] is the head loss at WE.
Kc = 0.5 unless given otherwise.
ABS = the absolute value.

Wall Exit (Sudden Expansion)

Energy loss is a wall exit = 1.0 ABS [HV2 - HV1]

In WX find D1 or D2 by iteration in the following equation:

D2 + HV2 + 1.0 ABS [HV2 - HV1] = D1 + HV1

Processing 'Error' and 'Warning' Messages

Error Messages - Editing the 'Element Cards':

THE ABOVE SYSTEM OUTLET WAS FOUND TO BE IN ERROR - ELEMENT NOT EQUAL TO 001

The *System Outlet* must be the first element to be processed. Check order of input to make sure the *System Outlet* card follows the title cards.

THE ABOVE INPUT CARD DID NOT CONTAIN THE REQUIRED DATA XXX

Check data on the input card with the input documentation for that element to make sure all the required data is present.

THE ABOVE INPUT CARD CONTAINED AN INVALID ELEMENT NUMBER

Invalid code in the element type field.

THE ABOVE INPUT CARD CONTAINED AN INVALID STATION

Station is not in sequence with previous stations. Value is less than the station of the previous element.

DURING EDIT PHASE XXX ERRORS WERE ENCOUNTERED - PROCESSING WILL NOT START

The number of errors in the edit phase is shown (XXX). Calculations will not begin until all edit errors are corrected.

NO EDIT ERRORS ENCOUNTERED - COMPUTATION IS NOW BEGINNING

All data passes edit checks, and processing calculations will begin.

A BLANK INVERT WAS GIVEN ON AN ELEMENT CARD

Card requires an invert, but none was entered.

INVALID SECTION NUMBER ON ELEMENT CARD

Section number must be between 1 and ZOO.

SECTION NUMBER HAD NO DATA FOR CHANNEL DEFINITION

The section number on the element card refers to a section that was not defined, or that was labeled as being in error during editing of the channel cards.

SECTION NUMBER HAD NO DATA FOR CROSS SECTION

Same as Message #10, only for cross-section data instead of channel definition data.

THE CHANNEL DEFINITION REFERENCED DID NOT CONTAIN THE REQUIRED DATA TO BE USED IN THIS ELEMENT

There is a conflict **between** data in the channel definition used to describe this element, and the type of element being described. Check restrictions for this element and make sure the channel definition selected has applicable data.

THE PREVIOUS SECTION OR CHANNEL DEFINITION DID NOT COINCIDE WITH THE DATA UTILIZED IN THIS ELEMENT

There is a conflict between data in the channel definition of the previous element and the current element being used in the element being described. Check restrictions for this element type and the channel definition data used.

Warning Messages - Editing the 'Element Cards':

THE ABOVE ELEMENT CONTAINED AN INVERT ELEV WHICH WAS NOT GREATER THAN THE PREVIOUS INVERT ELEV

Check inverts on the preceding and the current elements to make sure they are what you want. Program assumes data is good and will continue.

WARNING - ADJACENT SECTIONS ARE NOT IDENTICAL - SEE SECTION NUMBERS AND CHANNEL DEFINITIONS

The two adjacent sections are supposed to be identical, but if only the channel height varies it is allright. Check channel definitions to see if data is correct and only height varies. Program assumes data is good and will continue.

WARNING - PREVIOUS SECTION NUMBER WAS INVALID OR 0 - SEE PREVIOUS DESCRIPTION

Previous element should have been flagged as being bad, so the data passed to this element is zeroes. Processing for the other elements continues.

Error Messages - Sequence Checking thru Channel Definition Data:

NO SYSTEM HEADWORKS CARD - CANNOT TELL WHERE THE START OF CHANNEL DEFINITION DATA IS - NO PROCESSING

There must be a *System Headworks* card at the end of the element cards, just preceding the channel definition cards.

CHANNEL DEFINITION DATA (CD) DID NOT FOLLOW THE SYSTEMS HEADWORKS CARD - CONTINUING TO LOOK FOR CD OR **'PTS**

There must be at least one CD card following the *System Headworks* card, and all CD cards follow the *System Headworks* and come before the cross-section points (PTS) cards.

NO CHANNEL DEFINITION (CD) OR CROSS SECTION POINT CARDS (PTS) WERE RECOGNIZED - CHECK DATA

There must be at least one channel definition card following the *System Headworks* card. Check input data card code columns.

NO CHANNEL DEFINITION CARDS BEFORE CROSS SECTION POINT CARDS - CHECK DATA

Check order of input cards. Element cards ending with *System Headworks* must be followed by at least one channel definition card. Cross-section point cards follow the channel definition cards.

INVALID CHANNEL TYPE ON CHANNEL DEFINITION CARD - ITYPE = X SECT = XXX

ITYPE is the channel type requested and SECT is the section number the channel type is specified to define. Channel type must be a number behveen 1 and 6.

NO CROSS SECTION POINTS ENCOUNTERED - ASSUME NO IRREGULAR CHANNELS

No irregular channels or cross-section points are indicated for this problem. This is a warning message. Processing will continue.

INVALID CARD CODE ENCOUNTERED WHILE PROCESSING CD AND PTS CARDS CODE = XXX

After the first CD card, a card was found which did not have a code of CD or PTS. CODE indicates the invalid card code which should be corrected or placed in the correct order.

NO SYSTEM HEADWORKS CARD BEFORE CHANNEL DEFINITION OR CROSS SECTION POINTS

The *System Headworks* card was omitted or is out of sequence. It should be the last element card and should immediately precede the channel definition cards.

Error Messages - Sequence Checking Cross-section Points Cards:

INVALID OR MISSING NUMBER OF POINTS VALUE - MUST BE BETWEEN 3 AND 99 CODE = XXX ISECT = XXX NO PTS = XX

The number of points value is in error, or the card is out of sequence. This is supposed to be the first card of a cross-section for the section points. CODE is the card code. ISECT is the section number, and NO PTS is the number of points indicated.

INVALID CARD CODE FOR CROSS SECTION POINTS

While processing PTS cards, a code not equal to PTS was found. Code is incorrect, or card is out of sequence.

STARTED ANOTHER CROSS SECTION GROUP BEFORE PREVIOUS GROUP WAS COMPLETED

A new cross-section group was indicated (number of points was given) before all the points indicated by the previous number of points were read. Check card sequencing and make sure the number of points is only on the first card of a section, and is correct with the number of points to be read.

SECTION NUMBER IS INVALID OR MISSING - MUST BE BETWEEN 1 AND 200 CODE = XXX ISECT = XXX NO PTS = XXX

The section number entered is in error. CODE is the card code, ISECT is the invalid section number, and NO PTS is the number of points.

END OF FILE BEFORE ALL POINTS WERE READ ON IAST CROSS SECTION

The last input card was read before all the points indicated to exist in the current cross-section were read. Supply the remaining cross-section points cards to complete the section, or correct the number of points indicated to define the section.

END OF FILE ON CROSS SECTION POINTS

The last cross-section points card was read and processing in this program is completed.

NO CHANNEL DEFINITION RECORD FOR THIS SECTION CODE - GOING ON TO NEXT CROSS SECTION SECT = XXX

There was no channel definition, or an invalid channel definition was given at the corresponding section number, so no processing was done on these cross-section points.

MISSING NUMBER OF POINTS FOR CODE XXX FOR SECTION XXX

The first card to describe the cross-section points of a section did not have the number of points value to indicate how many points are to be read to describe the channel. Make sure that this is supposed to be the first card of the section points, and that the correct number of points value for this section is entered. If it is not supposed to be the first card of the section points, put cards in proper sequence.

Error Messages - Channel Definition Processing:

SECTION NUMBER INVALID OR MISSING, DATA CANNOT BE WRITTEN TO THE OUTPUT FILE

There is an invalid section number on a CD or PTS card. Section number must be between 1 and 200.

INVALID VALUE FOR THE NUMBER OF PIERS - MUST BE BETWEEN 0 AND 10 IF GIVEN

The number of piers on the CD card is invalid. Must be between zero and ten.

AVERAGE WIDTH OF PIERS IS INVALID OR NOT GIVEN WHEN THERE IS A VALUE FOR

NUMBER OF PIERS IN THE CHANNEL.

When some number of piers is entered, a value for the average width of the piers must also be entered. Correct either number of piers entry or average pier width entry.

CHANNEL HEIGHT IS INVALID OR IS NOT GIVEN

Correct height data entry in the channel definition.

CHANNEL DIAMETER IS INVALID OR IS NOT GIVEN

Correct the diameter data entry in the channel definition.

CHANNEL WIDTH IS INVALID OR IS NOT GIVEN

Correct the width data entry in the channel definition.

THERE IS A DIFFERENCE BETWEEN THE NO. OF PIERS AND THE NUMBER OF VALUES FOR PIER DEPTHS

If fewer depths are entered for piers in an irregular section than than the number of piers indicated, the remaining pier base values must be added, even if they are zero. If more depths are entered, only the amount up to the number of piers declared will be considered.

Error Messages - Cross-section Point Processing:

ENCOUNTERED A POINT WHERE X = 0 AND Y = 0 BEFORE ALL THE INDICATED POINTS WERE PROCESSED - ASSUMING ERROR

Only the first coordinate of the cross-section points can be 0, 0. Otherwise, the program cannot distinguish between blanks and zeros. If point desired is 0, 0 use .01,.01 for approximate data.

THE CROSS SECTION POINTS ARE OUT OF SEQUENCE FOR AN IRREGULAR OPEN SECTION - MUST BE COUNTERCLOCKWISE FROM MINIMUM X

Check the sequence of points on the cross-section point cards for the data which is out of order.

THE CROSS SECTION POINTS ARE OUT OF SEQUENCE FOR AN IRREGULAR SECTION - MUST BE COUNTERCLOCKWISE FROM MINIMUM X

Check the sequence of points on the cross-section point cards for data that is out of order. When maximum X is reached, the following X values must continually decrease.

MAXIMUM Y IS NOT AT EITHER SIDE OF AN OPEN IRREGULAR CHANNEL - ASSUMED BAD DATA AND PROCESSING IS STOPPED

For some reason, maximum Y was not at the end of an open irregular channel. Check input data and correct.

Computation Error and Warning Messages:

WATER SURFACE ELEVATION GIVEN IS LESS THAN OR EQUALS INVERT ELEVATION IN XXX, W.S. ELEV = INV + DC

The subroutine name is shown. This is a warning message that there was no water surface elevation entered for either the headworks or outlet, or that the water surface entered is less than the invert elevation causing DC to be the controlling depth. Processing continues.

WENTDS, NO AREA OF OBSTRUCTION IN ELEMENT XXX, Al = XXX, A2 =XXX

The element number, area in the upstream end, and area in the downstream end (based on depth from the upstream end) are printed. The area in the upstream end must be greater than the area in the downstream end. Make sure this is supposed to be a *Wall Entrance* and that the channel sections are described properly. Processing is stopped.

W.S. ELEV IS 10 FEET OR MORE ABOVE OPEN CHANNEL WALLS IN XXX, STATION = XXX, D = XXX, DH = XXX

The subroutine, station, depth, and maximum open flow depth are shown. Open flow depth reached the maximum limit in the program. Raise the heights of the channel walls at this point and rerun. Processing is stopped.

OVER 50 RECORDS WRITTEN IN XXX. ELEMENT = XXX, STATION = XXX

The subroutine, element, and station are shown. The maximum number of 50 intermediate points in a *Reach* element have been processed. Divide this *Reach* element into two or more *Reaches* at the station shown, and rerun. Processing is stopped.

CANNOT SOLVE QUADRATIC FORMULA FOR START OF OPEN FLOW IN RCHUS, STATION = XXX

The station at the downstream end of the *Reach* is shown. The solution to solving the quadratic formula was negative for the length of *Reach* in pressure flow. There is no solution for this problem. This situation should not occur, but if it does, this element must be hand calculated, and the other elements can be run with the hand calculated control depths. Processing is stopped.

THE KNOWN DEPTH EQUALED THE NORMAL DEPTH IN BERNLI, DEPTH = XXX

The known depth is shown. This is a warning message that normal depth has already been reached. The depth at the end of the *Reach* is set equal to normal depth. Processing continues.

THE UPPER AND LOWER LIMIT VALUES CALCULATED IN BERNLI WERE THE SAME, LOWER LIMIT = XXX, UPPER LIMIT = XXX

The values from ***Bernoulli's Equation*** based on the lower and upper limit depths are shown. This is a warning message indicating that depth cannot be found by ***Bernoulli's Equation,*** and that the upper and lower limit depths are the same. The depth at the end of the ***Reach*** is set equal to the current known depth. Processing continues.

THE VALUE TO SOLVE FOR DEPTH IN BERNLI IS NOT BETWEEN THE UPPER AND LOWER VALUE LIMITS, DESIRED VALUE = XXX, UPPER LIMIT VALUE = XXX, LOWER LIMIT VALUE = XXX, UPPER LIMIT DEPTH = XXX, LOWER LIMIT DEPTH = XXX

The value needed to solve ***Bernoulli's Equation,*** the upper and lower limit values from ***Bernoulli's Equation,*** and the upper and lower limit depths are shown. This is a warning message indicating that depth to solve *Bernoulli's* ***Equation*** cannot be found between the limits where it is expected. Depth at the end of the *Reach* is set to the current known depth, or to normal depth depending on whether the desired value to solve ***Bernoulli's Equation*** is greater or less than the prescribed limits. Processing continues.

THE XX FILE DOES NOT HAVE DEPTH AT THE HYDRAULIC JUMP IN JUMPR

The upstream or downstream file is shown. The station of the hydraulic jump cannot be computed, although it is indicated to exist because the upstream and downstream force curves crossed. Processing is stopped.

NO INTERSECTION OF FORCE CURVES COULD BE FOUND FOR THE HYDRAULIC JUMP IN JUMPR

A hydraulic jump was indicated, but there was insufficient data in the upstream and downstream files to locate the point of intersection. Processing is stopped.

THE FORCE AT THE HYDRAULIC JUMP IS NOT BETWEEN THE FORCES FROM THE UPPER AND LOWER LIMIT DEPTHS. UPPER LIMIT DEPTH = XXX, LOWER LIMIT DEPTH = XXX, UPPER LIMIT FORCE = XXX, LOWER LIMIT FORCE = XXX, FORCE AT JUMP = XXX IN PPMDEP

The upper and lower limit depths (depth from either side of indicated hydraulic jump), the upper and lower limit forces, and the force at the hydraulic jump are shown. The force at the jump should be equal or between the forces on either side of the jump, but this was not the case. Either the force entered for the jump or the points entered from the upstream and downstream file adjacent to the jump are wrong. Check upstream and downstream files for valid data. Processing is stopped.

THE TEST DEPTH EXCEEDED THE UPPER LIMIT DEPTH BEFORE THE FORCE AT THE JUMP WAS REACHED. TEST DEPTH = XXX, UPPER LIMIT DEPTH = XXX, TEST FORCE = XXX, JUMP FORCE = XXX IN PPMDEP

The iterated depth, upper limit depth, iterated force, and force at the hydraulic jump are shown. The depth causing the force at the hydraulic jump should be equal or between the depths on either side of the jump, but this was not the case. Either the force entered for the jump or the points entered from the upstream and downstream file adjacent to the jump are wrong. Check the upstream and downstream files for valid data. Processing is stopped.

XXX ERRORS WERE ENCOUNTERED IN SETTING THE PRELIMINARY VALUES IN ELMCHG

The number of errors in analyzing adjacent elements and flow rates and computing critical and normal depths are shown. These errors must be corrected, and the program must be rerun before actual processing will start. Processing is stopped.

NO XX RECORDS EXISTED WHERE INDICATED - ELEMENT NO. XXX IN WRITEN

The upstream or downstream file and the element number are shown. The upstream or downstream processing code indicated the computation for the element was valid, but there were no records in that file for the element. Processing continues with the next element.

THERE WAS NO JUMP INDICATED WHEN BOTH U/S AND D/S RECORDS EXISTED FOR ELEMENT XXX IN WRITEN

The element number is shown. There was a problem in the jump processing for this element. Either one of the profiles should be deleted or a hydraulic jump should be indicated. Processing continues with the next element.

A JUMP WAS INDICATED BUT THERE WERE NOT RECORDS ON BOTH THE U/S AND D/S PROFILES FOR ELEMENT XXX IN WRITEN

The element number is shown. There was a problem in the jump processing for this element. If the entire upstream and downstream profile is deleted, then there cannot be a jump. If there is a jump, there must be upstream and downstream profile data. Processing continues with the next element.

THERE WERE NO RECORDS FOR ELEMENT XXX IN WRITEN

The element number is shown. This is a warning message to indicate that there was no upstream or downstream processing for this element. Check the upstream and downstream profiles to verily this. If there is data, there is an internal problem. If there is no data, check the construction of the element. Processing continues with the next element.

NO PLOT GENERATED, BAD DATA OR NOT ENOUGH POINTS, 3 OR LESS

If there are only three elements being run, no plot will be generated. Processing will continue.

ELEMENT NUMBER XXX HAS ADJACENT ELEMENTS WHICH ARE IN ERROR

The element number is shown. There is an error in the sequence of elements (such as *Bridge Exits* back to back), which are not allowed. Check sequence of the elements, correct the error, and rerun. Sequence checking will continue, but processing will be stopped.

XXX DEPTH COULD NOT BE FOUND IN ELEMENT XXX

Either normal or critical depth and element number are printed. There is either an error in function DCRIT or DNORM or there is a bad channel description. Hand calculate the value, and if it is valid for the channel, try rerunning the program. Elements will continue to be checked, but no processing will occur.

IRREGULAR XXX VALUES ARE ZERO OR NEGATIVE, SET XXX EQUAL TO ZERO, XXX = XXX, PIERXXX = XXX, INXXX

Either force, area, or wetted perimeter values are shown from functions FORCEI, AREACI, or WETPI for irregular sections. The appropriate data could not be computed in this irregular section. Problem is internal to program.

PIER WIDTH IS WIDER THAN CHANNEL WIDTH IN XXX, DEPTH = XXX, PIER WIDTH = XXX

Either force, area, or wetted perimeter, depth, and average pier width is shown. The width of the number of piers at the given depth is wider than the channel width at that depth. Correct the entered data and rerun.

DEPTH EXCEEDS XXX WITH FORCE TOO LOW IN FORCEM. TEST DEPTH = XXX, TEST FORCE = XXX, XXX = XXX, DESIRED FORCE = XXX

The iterated depth and force, the maximum or minimum depth, and the desired force value are shown. The desired force in the *Bridge Exit* could not be reached within the prescribed depth limits. The desired depth is set to zero and no processing is done in that end of the *Bridge Exit.* Processing continues with the next element.

DESIRED FORCE IS OUTSIDE THE RANGE OF DEPTHS IN FORCEF. TEST DEPTH = XXX, TEST FORCE = XXX, XXX = XXX, DESIRED FORCE = XXX

The iterated depth and force, the maximum or minimum depth, and the desired force are shown. In downstream processing the desired force in the downstream end of the *Bridge Entrance* could not be reached within the prescribed limits of depth, so the desired depth is set to zero and no computation is done in the downstream end. In upstream processing the *Bridge Entrance* was under pressure at the upstream end, so pressure flow calculations will be done. Processing is continued for pressure flow going upstream and in the next element going downstream.

DESIRED FORCE IS OUT THE RANGE OF DEPTHS IN FWALL. TEST DEPTH = XXX, TEST FORCE = XXX, MINIMUM DEPTH = XXX, DESIRED FORCE = XXX

Same as previous message, except for *Wall Entrance* rather than *Bridge Entrance*.

DEPTH IS OUTSIDE THE RANGE OF THE POINTS DESCRIBING THE CHANNEL IN XXX. DEPTH = XXX, YMIN = XXX, YMAX = XXX

Either force, area, or wetted perimeter values for depth and minimum and maximum Y values are shown. If the depth is not above maximum open flow depth, there is an internal error. If the depth exceeds maximum open flow depth, raise the channel walls. Processing will continue, but errors should be corrected and program rerun.

UNABLE TO CALCULATE FRICTION SLOPE WITH MANNINGS EQUATION IN SF. AREA = XXX, WETTED PERIMETER = XXX

The area and wetted perimeter are shown. Either the area or the wetted perimeter should be less than or equal to zero. Processing is stopped.

CRITICAL DEPTH MAY BE INACCURATE IN ELEMENT XXX. INCREMENT = XXX

The element number and increment value are shown. If the increment is large, then critical depth is probably above the top of the channel, but is set equal to the channel height. If the increment is small, critical depth is probably pretty accurate but for some reason cannot be computed precisely. Processing continues.

Q VALUES IN THE JUNCTION ARE INCORRECT FOR DEPSMP. Q1 = XXX, Q2 = XXX, Q3 = XXX, Q4 = XXX

The Q (flow) values for both upstream and downstream ends and for the laterals are shown. Q2 should equal the sum of the other Q's. If it does not, there is an internal program problem. If the Q values are in error, reenter Q values. Processing is stopped.

A LATERAL ANGLE OF CONFLUENCE IS GREATER THAN 90 DEGREES IN DEPSMP. FIRST ANGLE OF CONFLUENCE = XXX

The angles of the laterals are shown. Check the entered values for the lateral angles, and correct any errors. Angles must be in degrees. Processing is stopped.

INVALID PROCESSING CODE WAS ENCOUNTERED IN XXX. PROCESSING CODE = X, AND IT SHOULD BE 1 OR 2

Function DEPSMP, SUMM, or SUMP is shown, with the processing code. The processing code should be 1 for downstream processing and 2 for upstream processing, representing the known end of the element. This is a program problem, and processing is stopped.

MOMENTUM AND PRESSURE CURVES DID NOT CROSS IN DEPSMP, SETTING DEPTH EQUAL TO UPPER LIMIT DEPTH PLUS ONE FOOT. DEPTH = XXX

The depth is shown. The intersection of the pressure and momentum curves was above the maximum open flow depth. For a closed channel, pressure flow calcula-

tions will be executed. Otherwise, processing will stop because depth is too high in an open channel.

(This page intentionally left blank)

Section 5

General Hydraulics

This level is used to calculate either the flow **capacity or the amount of flow in the following types** of channels or hydraulic devices.

Irregular-Shaped channel, with up to 3 flow lines. (see Notes 1 & 3)
Trapezoidal, Box, Pipe, and Weir structures. (see Notes 1 & 2)
Street flow without inlets.
Street flow with Street Inlet Analysis (curb, grate, and slot inlets).
Pump/Turbine **power analysis.**

Note 1: Flow rates at multiple depths optional.
Note 2: Multiple structures at various depths optional.
Note 3: Depths may be determined from multiple flow rates.

Program will accept *English* or *Metric* units. It is accessed by entering [5] from the *Hydrology/Hydraulics Menu (see Overview). The* CRT will then display:

```
GENERAL PROGRAM OPERATION:

Channel Analysis Program Options:
     1 - Create a new study file.
     2 - Run file, detailed report.
     3 - Change file entries.
     4 - None of the above, exit program.

                    Enter program option desired >
```

Enter [1] to begin a new analysis. If an analysis was previously done, you can enter [2] to create the output report, or enter [3] to edit previously entered data.

Note: You will be returned to the above menu after each study is completed. Then, to create a report file, enter [2] from the above menu.

If [1] is entered to begin a new study, the CRT will display the *Channel Design/Analysis Options Menu*, shown on the next page.

```
CHANNEL DESIGN/ANALYSIS PROGRAM OPTIONS:
Multiple flow/depth calculations permitted in options 1 & 2 below:

   1 - Irregular shaped channel, up to 3 flow lines.
   2 - Trapezoidal, Box, Pipe, Weir structures.
   3 - Street flow only.
   4 - Street flow + Street Inlet Analysis (curb; grate, slot).
   5 - Pump/Turbine power analysis.

              Enter the desired channel analysis option >

        Enter 0 if English Units; 1 if Metric (SI) Units >

      Enter 0 if flow rate (Q) is known, or 1 to find flow rate >
```

Enter the appropriate menu number for the type of analysis that you want to perform. Then enter [0] (zero) if you are working in *English* units, or [1] if in *Metric* units. (For the remainder of this section of documentation, we will assume that you are working in *English* units.)

Then enter [0] (zero) if you will be using a **known flow rate, or enter** [1] if you want the program to calculate flow rates. If you enter [0] to use a known flow rate, the CRT will display:

```
Enter maximum flow rate at headworks. (CFS) >
```

If [1] is entered to calculate flow rates, and if you are analyzing an *Irregular-Shaped* or *Street* channel (menu items 1, 3, or 4 above), the program requires a *depth of flow* entry. The CRT will then display the following instead of the above message.

```
Enter maximum depth (HGL) of flow at headworks (Ft.) >
```

Enter either the maximum depth of flow, in feet, or the maximum flow rate, in CFS, at the beginning of the channel.

All of the above analysis options require elevation and distance data entries. In the case of multiple channels it requires reference elevations and a distance from the system headworks to the system outlet (individual channels between these points may be longer or shorter). The CRT will display:

```
Enter (system headworks) invert elevation (Ft.) >
Enter (system outlet) invert elevation (Ft.)>
Enter Length or distance between the above points (Ft.) >
```

Enter the headworks and outlet elevations, in feet, and the distance between these two points, in feet.

Subsequent CRT entries depend upon the type of analysis being run, and each type is described in subsequent paragraphs.

5.1 - Irregular-Shaped Channels

Irregular Channel analysis calculates the steady state normal and critical depths of a given range of flow rates; or the flow rate for a given set of normal depths. Up to 100 flow depths or flow rate steps cay be calculated from zero up to the maximum depth or flow rate entered. The CRT will now ask for the ***Total number of steps desired.***

Irregular-shaped channels may have up to **3** flow lines, or three sub-channels, with each having up to 20 cross-section data points. A sub-channel is defined as any low point, with higher points on either side. The CRT will now ask for the ***Total number of subchannels in irregular channel.***

The overall channel is described by *X,Y* data points, where X represents the cumulative horizonatal distance (X coordinate) and Y represents the cumulative vertical (Y coordinate) cross-section distance. The beginning value of X is set ***to*** zero by the program. You can use any reasonable positive value of Y as the starting Y data coordinate.

If you have more than one flow line (sub-channel), the *X,Y* data points are entered separately for each sub-channel. The X value for the beginning of the first sub-channel will be zero. The first data point for the second sub-channel must be identical to the last data **point in the first sub-channel. And,** if there is a third sub-channel, its beginning data point must be identical to the last data point in the second sub-channel. You must determine the number of data points in each sub-channel. The ending point in sub-channel number 1 is counted for sub-channel number 1 and is also the **first** data point in sub-channel number 2. If there is a third sub-channel, the end point in sub-channel number 2 is also counted as the first point in sub-channel number 3.

The CRT will begin with sub-channel number 1, and will ask for the ***Total number of cross-section points*** in that channel. It will also ask for the ***Manning's 'N'*** friction factor for that sub-channel. You then will begin entering the *X,Y* coordinates for each data point in that sub-channel. When all points are entered for sub-channel number 1, the process is repeated for any subsequent sub-channels.

If there is **more than one flow line, remember to repeat the end point of the previous sub-channel for the start point of the next sub-channel.**

After all cross-section data points are entered, the program will calculate the normal steady state depth or flow rates. The critical depth and ***Froude*** numbers are also calculated.

5.2 - Trapezoidal, Box, Pipe, or Weir Structures

This section calculates the flow rates for up to 10 channel structures in either pressure or non-pressure flow through a range of up to 100 depth steps. It also calculates the depth of flow of any one type of channel structure.

Using Known Range of Depths: The program calculates flow rate versus inlet and outlet depths and creates a ***Flow Rate Summary Report*** of these calculations. This is useful in analyzing/designing one or more channels connecting two water surfaces, with either variable or fixed water surface elevations at the inlet and outlet ends of the channels. If more than one channel is used, channel inlet and outlet elevations of each channel can be different.

Using a detention basin as an example, you might have a 24 inch pipe at the bottom of the basin to handle low flow rates. You might then have weir or a box channel at some point above the pipe inlet to handle higher flows. By using a range of depths in the basin, you can calculate flow rate in each channel at each depth increment, and create a summary report of these flow rates. If the channels outlet into another basin, water surface at the outlet end can also be incremented from a minimum to a maximum surface elevation. Or it can be set at a fixed outlet water surface elevation.

Using Known Flow Rate: In this mode, you can only analyze one channel, using one flow rate. The program will analyze flow through a channel when channel size and flow rate is entered, calculating depth of flow, friction loss, and determines whether flow is pressure or non-pressure. For pipes, you can also calculate required pipe size for non-pressure flow (using d/D of .938) for the entered flow rate.

5.2.1 - Using Known Range of Depths

Enter [2] from the ***Channel Design/Analysis Options Menu*** to access this program level. Then select either ***English*** or ***Metric*** units. Then enter [1] to ***find flow rate. You*** must enter [1] or you will not be in the ***Using Known Range of Depths*** mode.

The CRT will then display:

```
Enter (system headworks) invert elevation (Ft.) >
Enter (system outlet) invert elevation (Ft.) >
Enter length or distancebetweenthe above points (Ft.) >
```

If you are only going to use one channel, enter its inlet (headworks) and outlet invert elevations, and then enter its length between those two points. If you are going to use more than one channel, the invert elevations become ***reference elevations*** for the system. You should normally enter the lowest inlet and outlet elevations for the system of channels. The distance entered should be the ***straight-line*** distance between those points. Actual channel lengths will be entered later for each channel.

The CRT will then display:

```
Enter system outlet water surface depth at MINIMUM flow (Ft.) >

Enter system outlet water surface depth at MAXIMUM flow (Ft.) >
```

These entries set the water surface criteria for the OUTLET end of the channel(s).

If the channels are ***free outlets*** (exiting to atmosphere), enter [0] (zero) for both minimum and maximum depths.

If the outlet water surface is constant and is above the ***outlet reference elevation,*** enter it depth in feet above the outlet reference elevation for both minimum and maximum depths.

If the outlet water surface is not constant and is to be incremented along with the inlet depths, enter the minimum depth above the ***outlet reference elevation*** (usually zero) and then the maximum depth in feet above the ***outlet reference elevation.***

The CRT will then ask for ***Total Number of Channels Desired. You*** can include any number from 1 to 10 channels.

After selecting the number of channels, the CRT will display:

```
DATA ENTRY FOR CHANNELS (Number 1 of X)
The following types of channel(s) are available:

1 - Weir, either trapezoidal or flat crested.
2 - Pressure/non-pressure pipe(s).
3 - Free outlet pipe (weir type flow).
4 - Trapezoidal or Box structure, pressure/non-pressure:

                              Enter type of channel desired>

Enter height of channel inlet above system headworks (Ft.) >
```

If you are using more than one channel, select the type of channel for the first channel to be processed. Then enter its height above the system ***inlet reference elevation*** in feet, or enter [0] if it is the same elevation.

Subsequent entries for each of the channel types are described below:

Weir, either trapezoidal or flat crested:

When a *Weir* channel is selected, the CRT will display:

```
CHANNEL CUTLET STRUCTURES:
Forueir flow, the following information is needed
   for the equation outflow=CL*(H^exp)
L = length of the spillway crest; C = discharge coeficient
C = 3.0 for a perfect weir flow
C = 2.63 for a rectangular weir with L>15 Ft.
Exp = exponent value, 1.5 for flat crested, 2.5 for v-notch
Caution: For V-notch weirs use L=1, C=2.57*tan(open angle/2)

        Enter the length of the weir spillway, L (Ft.)'?'

                          Enter the desired 'C' value>

Enter the exponent value, exp(1.5 flat crested, 2.5 v-notch)>
```

Enter the *length* of the spillway, the C value, and the *exponent* value for the type of weir that you are using.

If you are using more than one channel, the CRT will then return you to the *DATA ENTRY FOR CHANNELS Menu* to select the next type of channel to be used. If you are only using one channel, or if all channels have been selected, the CRT will display the message shown in the *Flow Rate vs. Depth Options* paragraph later in this section.

Pressure/non-pressure pipe(s):

If **you** are using more than one channel, one or more of them may have lengths invert elevations that are different from the previously entered *reference elevations and length*. When more than one channel is being used, the CRT will ask for the *Length of the Channel*, in feet, and the *Elevation Difference*, in feet between inlet and outlet of this pipe channel.

The CRT will then display:

```
        Enter Mannings N for the pipe(s) in the reach >

     Enter number of pipes to be used along this reach >

      Type of Pipe: 1 = Circular; 2 = Elliptical
                           Enter desired pipe option >
```

Enter *Manning's 'N'* value to be used for the pipe. Typical N values will also be displayed on the CRT. Then enter the number of pipes to be used in this *channel*, normally one. You can have more than one pipe in this channel, **but they must all be the same diameter and have the same inlet and outlet invert elevations. Then enter the** *Type of Pipe*, ie, [1] for circular or [2] for elliptical.

The CRT will then ask for the *Sum of 'K' Factors for Minor Losses Inside the Pipe*. 'K' factors for typical fittings, etc. will be displayed on the screen to help you determine this value.

If you are using a circular pipe, the CRT will ask you for the *Pipe Size*, in inches. If you are using an elliptical pipe, the CRT will ask for the inside vertical and inside horizontal dimensions, in inches.

If you are using more than one channel, the CRT will then return you to the *DATA ENTRY FOR CHANNELS Menu* to select the next type of channel to be used. If you are only using one channel, or if all channels have been selected, the CRT will display the message shown in the *Flow Rate vs. Depth Options* paragraph later in this section.

Free outlet pipe (weir type flow):

A *Free Outlet Pipe* with *weir type flow* is defined as a pipe whose inlet lip is parallel to the water surface. For example, if a pipe is used at the low point in a detention pond to drain the pond, its longitudinal axis will be perpendicular to the water surface (straight down) for some distance, before angling to its eventual outlet. Thus its inlet will act as a wier. This wier-type pipe option should only be used for this type of circumstance.

The program limits the maximum outflow capacity to that obtained when the depth of flow over the top of the pipe equals the diameter of the pipe. If you are evaluating pipes with greater depths of flow over the inlet, the pressure/non-pressure pipe option should be used.

When a *Free Outlet Pipe*, with a weir type inlet, channel is selected, the CRT will ask for the *Pipe Diameter*. Then, if you are using more than one channel, the CRT will then return you to the *DATA ENTRY FOR CHANNELS Menu* to select the next type of channel to be used. If you are only using one channel, or if all channels have been selected, the CRT will display the message shown in the *Flow Rate vs. Depth Options* paragraph later in this section.

Trapezoidal or Box structure, pressure/non-pressure:

If you are using more than one channel, one or more of them may have lengths invert elevations that are different from the previously entered *reference elevations and length*. When more than one channel is being used, the CRT will ask for the *Length of the Channel*, in feet, and the *Elevation Difference*, in feet between inlet and outlet of this trapezoidal or box channel.

The CRT will then display:

```
IMPROVED CHANNEL DATA ENTRY:
Data entry for trapezoidal channel, rectangular channel
    or 'V'-shaped channel

        Enter mannings friction factor for the channel
                    Note: For improved concrete use n = .015 >

                  Enter horizontal base width of channel (Ft.) >

        Enter the maximum allowable depth of the channel (Ft.) >

             Enter 0 if open channel;
  Enter 1 if box or covered channel (pressure flow condition) >
```

Enter *Manning's 'N'* value for the type of channel surface you're using. Then enter the base width of the channel, in feet (enter [0] for a *V-Shaped* channel). Then enter the maximum depth, if feet. This will limit the depth of flow to the entered depth, and establishes the channel *cover* for box or covered channels. Then enter [0] (zero) for an open channel, or [1] for a box or covered channel.

The CRT will then display:

```
IMPROVED CHANNEL DATA ENTRY:
Data entry of channel side slopes or 'z' factors
    and number of channels this same size:

        Enter the Left channel bank 'z' factor (horiz/vert)>

       Enter the right channel bank 'z' factor (horiz/vert)>

                    Enter the total number of channels used >
```

The first two entries set the channel side-slope criteria. For a box channel, enter [0] (zero) for both entries. You can use more than one identical channel for this particular channel, but they all must be the same size. Enter the total number of channels that make up this particular channel (normally one).

The CRT will then ask for the *Sum of 'K' Factors for Minor Losses Inside the Box*. It will also display typical minor loss *K* factors for your use.

If you are using more than one channel, the CRT will then return you to the *DATA ENTRY FOR CHANNELS Menu* to select the next type of channel to be used. If you are only using one channel, or if all channels have been selected, the CRT will display the message shown in the *Flow Rate vs. Depth Options* paragraph below.

Flow Rate vs. Depth Options:

After all channels have been selected and data entered for each, the CRT will display the following message:

```
FLOW   RATE  VS.  DEPTH  OPTIONS:
The following options are available for calculating
either the depth of f Low or flow rate:
1 - Use only one inlet flow depth for the analysis.
2 - Use a range of inlet flow depths for computation.

Enter option desired>
```

A ***step analysis*** of flow rates at various depths can be performed, if desired, or you can use only one inlet flow depth. When either option is selected, the CRT will ask for the ***Initial depth of water at the system headworks,*** in feet. This is the depth **above the *beadworks reference elevation.***

If you enter [2] to use a range of inlet flow depths, the CRT will also as for the ***Maximum Depth of Water at System Headworks,*** in feet, and the ***Number of Steps of Depths between Min and Max Depth.*** For example, if you want the headworks water surface to vary from the ***reference elevation*** to 10 feet above the ***reference elevation, you*** would enter [0] (zero) for the ***Initial Depth*** and "10" for the ***Maximum Depth.*** Then, enter the ***Number of Steps*** desired between the minimum and maximum headworks water surface. For example, if you wanted flow rates calculated for each foot-depth of water surface, you would enter "10".

The program will then perform the flow rate calculations and return you to the ***General Program Operation Menu. You*** can then create a report file of the calculations, display them on the screen, edit the input data, do another study, or exit the program.

5.2.2 - Using Known Flow Rate

In this mode, you can only analyze one channel, using one flow rate. The program will analyze flow through a channel when channel size and flow rate is entered, calculating depth of flow, friction loss, and determines whether flow is pressure or non-pressure. For pipes, you can also calculate required pipe size for non-pressure flow **(using** d/D of .938) for the entered flow rate.

Enter [2] from the ***Channel Design/Analysis Options Menu*** to access this program level. Then select either ***English*** or ***Metric*** units. Then enter [0] (zero) to ***find depth of flow. You* must enter [0] or you will not be in the *Using Known Flow Rate*** mode.

The CRT will then ask for the ***Maximum Flow Rate at Headworks*** (system inlet), in CFS. After this entry, the CRT will display:

```
Enter (system headworks) invert elevation (Ft.) >
Enter (system outlet) invert elevation (Ft.) >
Enter length or distance between the above points (Ft.) >
```

Enter the channel inlet (headworks) and outlet invert elevations, and then enter its length between those two points. The CRT will then display:

```
DATA ENTRY FOR CHANNELS (Number 1 of X)
The following types of channel(s) are available:

1 - Weir, either trapezoidal or flat crested.
2 - Pressure/non-pressure pipe(s).
3 - Free outlet pipe (weir type flow).
4 - Trapezoidal or Box structure, pressure/non-pressure.

                              Enter type of channel desired >
```

Select the type of channel that you want to analyze. Then the CRT will ask for specific criteria entries, depending upon the type of channel selected. These entries are described in subparagraphs under paragraph 5.2.1 above, with the subparagraph titles as shown in the above menu, and will not be repeated here. The only difference will be for a pipe-type channel. In this case, the program will give you the option of entering a pipe size, or letting the program calculate a *required pipe size* to handle the entered flow rate under non-pressurized flow conditions.

5.3 - Street Flow Only

This option calculates the depth of flow or flow rate in a street cross-section. If the street inlet option is selected (see paragraph *5.4* below) the program will calculate the amount of street flow intercepted by either a curb inlet, grate inlet or slot inlet. Also, if the street inlet option is used, the program either sizes or evaluates the pipe connecting the street inlet to the storm drain main. If the entered connecting pipe size is too small, the the amount of flow intercepted is governed by the pipe capacity.

The first entries specific to street flow will be street cross-section data. The CRT will display a schematic of a *Typical Half-Street Cross-section* for your information and assistance in entering cross-section data.

If you have previously entered one or more street cross-sections, the CRT will allow you to select and use one of them, or to enter a different cross-section. If you are entering a street cross-section, the CRT wil ask for the following entries.

Note that the street section can include one **grade** break between the centerline and the edge-of-pavement (gutter lip).

Enter the **standard curb height, in inches.**

Then enter the **distance from the curb face to the property line,** in feet.

Then enter the **street half-width, in** feet (centerline to curb face).

Then enter the **distance from street centerline to grade break, in** feet. If there is no grade break between the centerline and the gutter lip, you still must enter some distance greater than zero. Then enter the same grade later in the program for both cross-falls.

Then enter the **gutter width,** in feet (curb face to gutter lip).

Then enter the **gutter hike, in** inches (difference between gutter flow line and lip of gutter).

Then enter the **decimal grade** (Vert/Horiz) from **curb to property line.** Do NOT enter as percent. If the slope is 5 percent, enter as ".05". Note: Positive grade shows property rises from curb.

Then enter the **decimal grade** (Vert/Horiz) from the **grade break to the street centerline. Note:** Positive grade shows centerline higher than grade break.

Then enter the decimal grade (Vert/Horiz) from edge-of-gutter (lip) to the grade break. Note: Positive grade shows break higher than gutter lip.

After the above entries are completed, the CRT will display the following:

```
Streetflow Options:

      1 - Run-off flows on one side of the street
      2 - Run-off flows evenly on both sides of the street

                          Select street flow option desired >
                      Enter Manning's N for gutter flow area >
                 Enter Manning's N for gutter to grade break >
                  Enter Manning's N for grade break to crown >
```

The above run-off options are available to determine the capacity or depth of flow in the street section that was described earlier. Enter the desired option, 1 or 2.

Then enter *Manning's 'N'* values for the paved street and the gutter sections. The program will calculate the depth of flow or capacity of the street and return you to the *General Program Operation menu.*

5.4 - Street Flow with Street Inlet Analysis

If the street with street inlet options was selected, the following data will be required, in addition to the entries described in paragraph *5.3* above.

```
NOTE: This program wilt calculate the amount of possibleflow
entering a street inlet using DOT-FHA HEC No. 12 formulae. After
themaximum inlet capacity is determined, the pipe capacity from
the street inlet(s) isdetermined and the lesser amount determines
the amount of flow remaining in the street at the inlet.

The following options are available:

   1 - Use the HEC-12 CURB inlet criteria for flow capacity.
   2 - Use the HEC-12 SLOT inlet criteria for flow capacity.
   3 - Use the HEC-12 GRATE inlet criteria for flow capacity.

                        Enter the inlet capacity option desired>

Pipe slope entry if different from street:
                        Enter the slope of the pipe (percent) >

                           Enter the length of the pipe (Ft.) >
```

Select and enter the type of inlet to be used.

To calculate the capacity of the pipe that connects the catch basin to the storm drain main, you must enter its slope, in percent. Note: If you *default* the displayed pipe slope entry or use the same slope as the street slope, the connecting pipe capacity will not be calculated. Then enter the length of the connecting pipe.

The CRT will then display:

```
The inlet formulas used require the total length of the inlet,
the height(or width) of opening, and the depth and width
of street depression from the normal street gutter flow line.
The inlet capacity is determined by formulas for street on a grade
if the street slope is greater than 0.5%; sag equations if less
than 0.5%

                              Enter the inlet length (Ft.) >

                  Enter the width of street depression (Ft.) >

                  Enter the depth of street depression (In.) >
```

Enter the length of the inlet opening, if feet. Then enter the width of the inlet *depression,* in feet. Then enter the depth of the inlet *depression,* in inches.

If you are using a curb or slot opening inlet, the CRT will then ask for the ***height of the curb inlet*** or the ***width of the slot opening.*** If you are using a grate inlet, it will ask for the ***width of the grate opening.*** All of these entries are in inches.

If you are using a ***grate type*** inlet, the CRT will display a schematic of a typical grate-opening inlet. It will then display eight types of grate covers for the inlet, as shown below.

```
The following types of grate inlet covers are available:
   1 - Parallel Bar, 1-7/8 in. o.c.
   2 - Parallel Bar, 1-7/8 in. o.c. with 3/8 in. diam. Lateral rod
   3 - Parallel Bar, 1-1/8 in. o.c. with spacers at 11-1/4 in.
   4 - Curved Vane, vanes 4-1/4 in. o.c.;longitl. bar 3-1/4 o.c.
  5 - 45 deg. Tilt bar at 4 in. o.c., long. bar at 3-1/4 in. o.c.
   6 - 45 deg. Tilt bar at 4 in. o.c., tong. bar at 2-1/4 in. o.c.
   7 - 30deg. Tilt bar at 4 in. o.c., Long. bar at 2-1/4 in. o.c.
  8 - Reticuline, 'Honeycomb', 1/4x4 in. long. bearing bars at
       2-9/16 in. o.c.; 3/16x2 in. reticuline bars, 3/8 in. rivits

                              Enter type of grate inlet >
```

Select and enter the type of grate inlet cover to be used.

If runoff flows on both sides of the street, the CRT will display the following:

```
The inlets may be Located on either one or both sides of the
street. If the inlet is Located on only one side of the street,.
themaximumpossible intercepted flow is 1/2 the current flow rate.
The following options are avai lable:

      1 - For one inlet on one side of the street.
      2 - For two inlets, one on each side of the street.

                         Enter the number of street inlets >
```

If runoff flows on both sides of the street, you can place an inlet on both sides, or only one side, of the street.

If you are calculating the capacity of the catch basin outlet pipe, the CRT will display:

```
          Enter Mannings N for the pipe(s) in the reach >

       Enter number of pipes to be used along this reach >

         Type of Pipe: 1 = Circular; 2 = Elliptical
                          Enter desired pipe option>
```

Enter ***Manning's 'N'*** value to be used for the pipe. Typical ***N*** values will also be displayed on the CRT. Then enter the number of pipes to be used for this ***connectingpipe,*** normally one. You can use more than one pipe, but they must all be the same diameter and have the same inlet and outlet invert elevations. Then enter the ***Type of Pipe,*** ie, circular or elliptical.

The CRT will then ask for the ***Sum of 'K' Factors for Minor Losses Inside the Pipe.*** 'K' factors for typical fittings, etc. will be displayed on the screen to help you determine this value.

If you are using a circular pipe, the CRT will ask you for the ***Pipe Size,*** in inches. If you are using an elliptical pipe, the CRT will ask for the inside vertical and inside horizontal dimensions, in inches.

The program will then perform the calculations and return you to the ***General Program Operation*** *menu.*

Inlet Calculations Formulae:

Curb Opening Inlets: The curb inlet capacity is calculated with the following formulas published by the U.S. Department of Transportation Federal Highway Administration (Hy-***draulic Engineering Circular No.*** *12).*

Length for total interception, Lt = .6 Q^.42 S^.3 (1/nSx)^.6
where Q = total flow, S = street slope, n = Mannings "n"
and Sx = equivalent cross slope into inlet
Efficiency, E = 1 - (1 - L/Lt)^1.8 , Where L = given length

The intercepted flow QI = QE

The program also calculates the maximum flow for a sump condition given the length and height of opening for determination of the maximum curb inlet flow with these equations:

Maximum Weir Flow (up to depth of opening)
QI = 2.3(L + 1.8W) d^1.5, where L = given length
W = 0 or gutter width and d= depth of flow at curb

Maximum Orifice Flow (If depth is higher than opening)
QI = .67 h L (2 g D0)^.5 , where h = opening width,
L = given length, g = 32.2, and DO = head on center
of the orifice throat.

Grate Inlets: The grate inlet capacity is calculated with the following formulas published by the U.S. Department of Transportation Federal Highway Administration ***(Hydraarlic Engineering Circular No.*** *12).*

Ratio of frontal flow to total gutter flow:
EO = Qw/Q = 1 - (1 - W/T)^2.67, where Q = total gutter flow,
Qw = flow in width W, W = width of depressed gutter
or grate, and T = Total spread of water.

Ratio of side flow, Qs, to total gutter flow is:

$Qs/Q = 1 - Qw/Q = 1 - EO$

Ratio of frontal flow intercepted to total frontal flow:
$Rf = 1 - 0.09 (V - VO)$ where V = velocity of gutter flow
VO = gutter velocity where splash over first occurs

Ratio of side flow intercepted to total side flow:
$Rs = 1 / (1 + 0.15V^{1.8} / Sx\, L^{2.3})$ where L = grate length
V = gutter velocity, Sx Cross slope

The efficiency, E, of a grate: $E = Rf\, EO + Rs(1 - EO)$
Total flow interception $Qi = E\, Q$, where Q = total gutter flow.

The maximum flow capacity is also compared with:
Orifice flow capacity = $.67A(2gd)^{.5}$, where A = clear opening area of the grate, g = 32.2 ft/s^2, d = depth of flow

Slot Inlet: The slot inlet capacity is calculated with the following formulas published by the U.S. Department of Transportation Federal Highway Administration ***(Hydraulic Engineering Circular No. 12).***

Length for total interception, $Lt = .6\, Q^{.42} S^{.3} (1/nSx)^{.6}$,
where Q = total flow, S = street slope, n = Mannings "n", and Sx = equivalent cross slope into inlet.

Efficiency, $E = 1 - (1 - L/Lt)^{1.8}$, where L = given length.

The intercepted flow QI = QE.

The program also calculates the maximum flow for a sump condition, given the length and width of opening, to determine the maximum slot inlet flow using the equations for orifice flow:

Maximum Orifice Flow (If depth is higher than width.),
$QI = .80\, L\, W\, (2\, g\, Dp)^{.5}$, where h = opening width,
L = given length, g = 32.2, and Dp = head on center of the orifice throat.

5.5 - Pump/Turbine Power Analysis

This program option calculates the power required of a pump for **a** given flow rate, or can calculate the flow **rate with a given horsepower of a pump.** For turbines, the program calculates either the flow rate required for a given power output, or the power output with a given flow rate. The program uses the *Bernoilli* equation, and considers the friction loss from the entered pipe(s), and uses the elevation difference and pressures enter for the inlet and outlet of the system.

When this option is selected, the CRT will display:

```
The program determines the f Lou using the difference in head
from the inlet and outlet elevations, pressures, and pipe losses,
then calculates the power required or generated by'the water flow.
...
Enter 1 if pump; or 2 if turbine >

Enter pressure or depth (Ft.) H2O at pipe inlet >

Enter pressure or depth (Ft.) H2O of water at pipe outlet >
```

Enter [1] if you are analyzing a pump, or [2] if a *turbine.* Then enter the pressure, or depth in *feet of water,* at the pipe inlet. Then enter the pressure, or depth in *feet of water,* at the pipe outlet.

If the flow rate is unknown, then you must also enter the power, in *horsepower,* of the pump/turbine.

The CRT will then display:

```
Enter Mannings N for the pipe(s) in the reach >

Enter number of pipes to be used along this reach >

Type of Pipe: 1 = Circular; 2 = Elliptical
Enter desired pipe option >
```

Enter *Manning's 'N'* value to be used for the pipe. Typical *N* values will also be displayed on the CRT. Then enter the number of pipes to be used for this analysis, normally one. You can use more than one connecting pipe, but they must all be the same diameter and have the same inlet and outlet invert elevations. Then enter the *Type of Pipe,* ie, circular or elliptical.
The CRT will then ask for the *Sum of 'K' Factors for Minor Losses Inside the Pipe.* 'K' factors for typical fittings, etc. will be displayed on the screen to help you determine this value.

If you are using a circular pipe, the CRT will ask you for the *Pipe Size,* in inches. If you are using an elliptical pipe, the CRT will ask for the inside vertical and inside horizontal dimensions, in inches.

The CRT will then ask for the efficiency of the pump/turbine being analyzed. If efficiency is 75 percent, enter ".75". If it is 100 percent, enter "1". Entries from ".01" to "1" are allowed.

The program will then perform the analysis and return you to the *General Program Operation menu.*

Section 6

Single Pipe - Pressure or Nonpressure Flow Calculations

This option is primarily used **to evaluate a single circular pipe in either open channel flow or pressure flow conditions. The** program calculates the depth of flow (or the head loss in the pressure flow option), or capacity, or size of pipe required. The open channel option includes elliptical pipe evaluations or size calculations.

For open channel flow the program uses the ***Manning's*** equation for calculations. When using the pressure flow option, the you can elect to have the program use ***Manning's*** equation, *Hazen-Williams* equation, or the *Darcy-Weisbach* equation (Default Moody diagram values of water at ***60*** degrees F).

You have the option of entering a viscosity and specific gravity of water at different temperatures or use data for other type fluids.

Options allow use of flow units in cubic feet per second (CFS), gallons per minute (GPM), or million gallons per day (MGD). When using the pressure pipe option, pressure units are either feet of water (FtH20) or pounds per square inch (PSI).

This level is accessed by entering [6] from the ***Hydrology/Hydraulics Menu.***

If you have previously run this program, the CRT will display:

```
An input file already exists for this program in
your account. The following options are available:

1 - Run the file.
2 - Change the file before running.
3 - Prepare a new data filer:---

DEFAULT exit to the DESIGN Menu.

Enter one of the above .....
```

If [2] or [3] is selected, or if you have not previously run this program, the CRT will display:

```
*** Pipe Flow Calculation Options .....
** OPEN CHANNEL FLOW ******************
1 - Calculate DEPTH OF FLOW, given:
Flowrate(Q), Pipe Size and Mannings "n"

2 - Calculate CAPACITY, given:
Depth of Flow, Pipe Size and Mannings "n"

3 - Calculate PIPE SIZE, given:
Flowrate(Q), Slope and Mannings "n"

** PRESSURE FLOW **********************

4 - Calculate PRESSURE, given:
Flowrate(Q), Pipe Size friction loss data

5 - Calculate CAPACITY, Q, given:
Pressure, Pipe Size and friction loss data

6 - Calculate PIPE SIZE, given:
Flowrate(Q), Pressure and friction loss data

Enter one of the above ....
```

Enter the menu number for the type of calculation that you want to perform, ie, enter [5] to calculate capacity when flowing under pressure.

The CRT will then ask for *Desired Flow Units.*

If you are running *Pressure Flow* calculations, the CRT will also display:

```
** PRESSURE UNITS **************
1 - Feet of water (FT H2O)
2 - Pounds per Square Inch (PSI)

Enter desired Pressure Units
```

Enter [1] if you are working with *Feet of water,* or [2] if *Pounds per Square Inch.*

Then enter the pipe invert elevation at both the pipe INLET and OUTLET, in feet.

Then enter the length of the pipe, in feet.

If you are running nonpressure flow calculations, the CRT will then ask for the *Depth of Flow* in the pipe.

If you are running pressure flow calculations, the CRT will ask for the *Head loss.* **NOTE:** The pressure difference entry is defined as the **HEAD LOSS** from pipe friction between the pipe inlet and outlet. The pressure difference from elevation change is added to the pipe friction losses by the program for user information.

If the pressure flow option was selected, the CRT will then display the following:

```
FRICTION LOSS EQUATION OPTION

1 - Use Mannings N equation
2 - Use Hazen-Williams equation
3 - Use Darcy-Weisbach equation (Moody Diagram)

Enter option desired
```

Enter the menu number for the type of equation that you want to use.

The CRT will then display friction factors for the selected type of equation and ask for your desired friction factor.

Then enter the ***Number of pipes to be used along this reach,*** and whether the pipe is circular or elliptical. The program calculates flow for up to 15 pipes that are connected in parallel, however, all pipes must be of the same size. The program calculates minor losses using the equation Hm = KV^2/2g for the total minor loss along the pipe(s).

If elliptical pipe(s) is being evaluated, the CRT will display a schematic of an elliptical pipe and ask you to enter data describing the horizontal to vertical ratio, or dimensions, of the elliptical pipe.

If you are running pressure flow calculations, a list of recommended minor loss 'K' factors will be displayed, and you will be asked to enter the SUM of the 'K' factors for the minor losses inside the pipe.

Then enter the diameter of the pipe(s), if you are calculating capacity, depth, or pressure.

The program then completes the calculations and displays them on the screen. You can then print the report on the ***default*** printer, or you can create an output file which can be reviewed in ***VUE*** or sent to a different printer by using the ***PRINT NAME =PIPE.OUT*** from system level, where NAME is the name of the printer you want to send it to.

(This page intentionally left blank)

Section 7

Sanitary Sewer Network

This program was designed for use with Gravity-flow Sanitary Sewer Systems, and is not recommended for Storm Sewer Systems that flow under pressure. It was developed using the guidelines outlined in ***Gravity Sanitary Sewer Design and Construction,*** ASCE Manuals and Reports on Engineering Practice No. 60, WPCF Manual of Practice No. FD-5.

The sewer system can have any number of manholes. The only limitation is a maximum of five lines coming into any one manhole. You cannot have more than five **lines** converging on **a** single manhole.

The Sewer Network program calculates sanitary sewage flow **from flow** = ***area times a*** **flow** ***constant*** and/or *flow* = ***capita times a*** *flow* ***constant.*** Infiltration flow is calculated using *flow* = ***area times a infiltration constant*** and/or *flow* = ***pipe diameter*** *per* ***unit length*** *times* ***a constant.***

Because of the wide variety of constants used throughout the United States (and the world), you must enter the constants for each particular sewer system study. Some examples of sewage and infiltration constants used in various areas can be found in the ***Help File*** that can be accessed from the Sewer ***Network Program Options*** menu, described later in this section.

For ease of converting the data from this program to a profile plot, the program calculates the pipe invert elevations entering and leaving a manhole, together with the invert elevation at the center of the manhole.

The program has two modes of operation. The ***Design Mode*** is used to assist you in designing a sewer network. It calculates and tabulates the total flow expected in each sewer line and the total sewer system being designed, and calculates the pipe sizes, slopes, and invert elevations required to maintain:

The desired minimum self cleansing velocity.
The required minimum pipe slope for each pipe size.
The minimum / maximum pipe depth below surface.

The Check Mode of operation evaluates an existing sewer network and determines pipe capacities (depth of flow) throughout the existing system. This mode can also be used to change or modify a *Design Mode* data file to fit particular dimensions or requirements.

Program Operation

The program is accessed by entering [7] from the ***Hydrology/Hydraulics Menu. The*** CRT will then display:

```
1 - WILD Nex File
2 - RUN Data File, Standard Detailed Report
3 - RUN Data File, Summary Form Report
4 - CORRECT Data File
5 - ADD TO Data File
6 - EXIT PROGRAM, return to menu
7 - NONE, VIEW OR PRINT HELP FILE
```

Menu item **#1** must always be used to **begin** the design of a sewer system or to check an existing sewer system. You will then be asked to enter a study NAME.

You should have a layout of the sewer system. We recommend that each manhole be assigned a manhole number and so annotated on the layout. If you plan to create profile data **files** from the output of this program, manhole numbers are required. Do not repeat manhole numbers. Each manhole must have a unique number.

If you do not elect to enter manhole numbers, you must enter the station value for each manhole, and those values should be annotated on your layout.

If manhole numbers are used, you have the option of entering either the station value at each manhole, or the distance between each manhole. We strongly recommend **using** manhole numbers so that profile **data files can be easily created** from this program's output. We also recommend that you use the *Distance Between Manholes* option, rather than the station option. By using the *Distance Between Manholes* option, any stationing datum can be used when the profile data files are created.

If your sewer layout sketch is complete, you are ready to begin building the sewer system data file. The general steps in building a new data file are as follows:

Select the desired 'Operation Mode', ie, either 'Design' or 'Check'. The 'Design' mode will be described first.

7.1 - Design Mode

General Data Entry:

For manhole data entry, you can enter the manhole station without manhole numbers, or enter manhole numbers and distances between manholes, or enter manhole numbers and the station for each manhole. If you select the first option, sewer profiles cannot be created from the output files.

You then have the option to enter data using *English* or *Metric* units. The profile programs do not support metric units at this time.

Then enter the *Types of Development* for the system. Up to 20 different *development types* can be used, and each will contain specific data for sewage and infiltration quantities. For example, when *English* units are used, sewage flow for each *development type* can be based upon gallons per day per acre, or gallons per day per capita. For infiltration, each ***development type*** can be based upon gallons per day per acre, or gallons per day per diameter inch per mile.

Then enter the total number of pipe sizes that you want considered in this sewer system design. Then enter each desired pipe size, in order from the smallest diameter to largest diameter. Normally sewer pipes range from ***6*** inches (8 inches is the smallest allowed by some agencies) to 10, 12, 15 and at **3** inch increments up to the largest size. The program considers your entered pipe listing as the only pipes available for normal lines (siphons and pumps are an exception) and **will** select the smallest size adequate to carry the flow when in the ***Design Mode.***

Then enter the total number of manhole sizes that will be used in the design, and the manhole inside diameter for each.

The program will then ask for the minimum desired flow velocity and the ***d/D (flow*** depth divided by pipe diameter) for pipes of 15 inches or less in diameter and for pipes greater than 15 inches in diameter. Standard ***default*** values will be displayed for each entry.

Then select the method of aligning different size pipes, either by matching **soffits** (inside top-of-pipe) or by matching inverts. Then enter the minimum cross-manhole drop, if one is required by your using agency.

Then enter the minimum invert depth below top-of-manhole, and the desired maximum invert depth below top-of-manhole.

Sewer line segments with low flow rates may require steep slopes to maintain minimum cleansing velocity, which can cause excessively deep lines and trenching. Some agencies allow overriding criteria, such as holding a slope of 0.0033 or using one-half the desired minimum velocity, for those low-flow lines. The program will now give you the option of maintaining minimum velocity for all lines, or using one of the other criteria for low-flow lines.

The CRT will now display all of the entered ***General Data. You*** should review the displayed data to make sure it is correct before continuing. If it is not correct, enter [2] and you will be returned to the program operation screen. Then proceed through the program. Your entered values or selections will be displayed as ***defaults.*** Change those that are incorrect, defaulting [RTN] those that are O.K. When all of the ***General Data*** entries are correct, continue by entering [1], and the CRT will display an abbreviated Sewer ***Flow Process Options*** menu, as shown below:

```
1 - BEGINNING of sewer line (no external flow at beginning
      point   of   line)

6 - USER SPECIFIED entry/override of flow in a line
```

Sewer Line Data Entry:

Figure 11-l will be used as an example to explain the remaining data entries. Each manhole is numbered along with the top-of-manhole elevation. The distance between each manhole is also shown, in feet. The outlet, or connection to an existing sewer line, is at MH #7.

Note: Top-of-manhole elevations are required. Before you can design a sewer system, you must have established the surface profile above the sewer lines. If the job includes street design, you should have designed the proposed street centerline profiles before starting the sewer design. If the street, or other surface above the sewer lines, is existing, you should have created existing surface profiles above the proposed sewer lines. In either case, you can then determine the top-of-manhole elevations before accessing this program.

You could start your data entry at MH #1, MH #8, MH #11 or MH #14. For this example, we will start at MH #1. As the above CRT display shows, you have two options for a ***beginning*** manhole (initial area) on a line. If the beginning manhole is the true beginning of the line and has **no external flow** from another sewer system that is not included in this study, you will select Option #1. If the beginning manhole is actually an existing manhole **that has a known flow** that will be diverted into your study system, you will select Option #6. One of these two options would be used for all four of the ***beginning*** manholes (1, 8, 11, and ***14).***

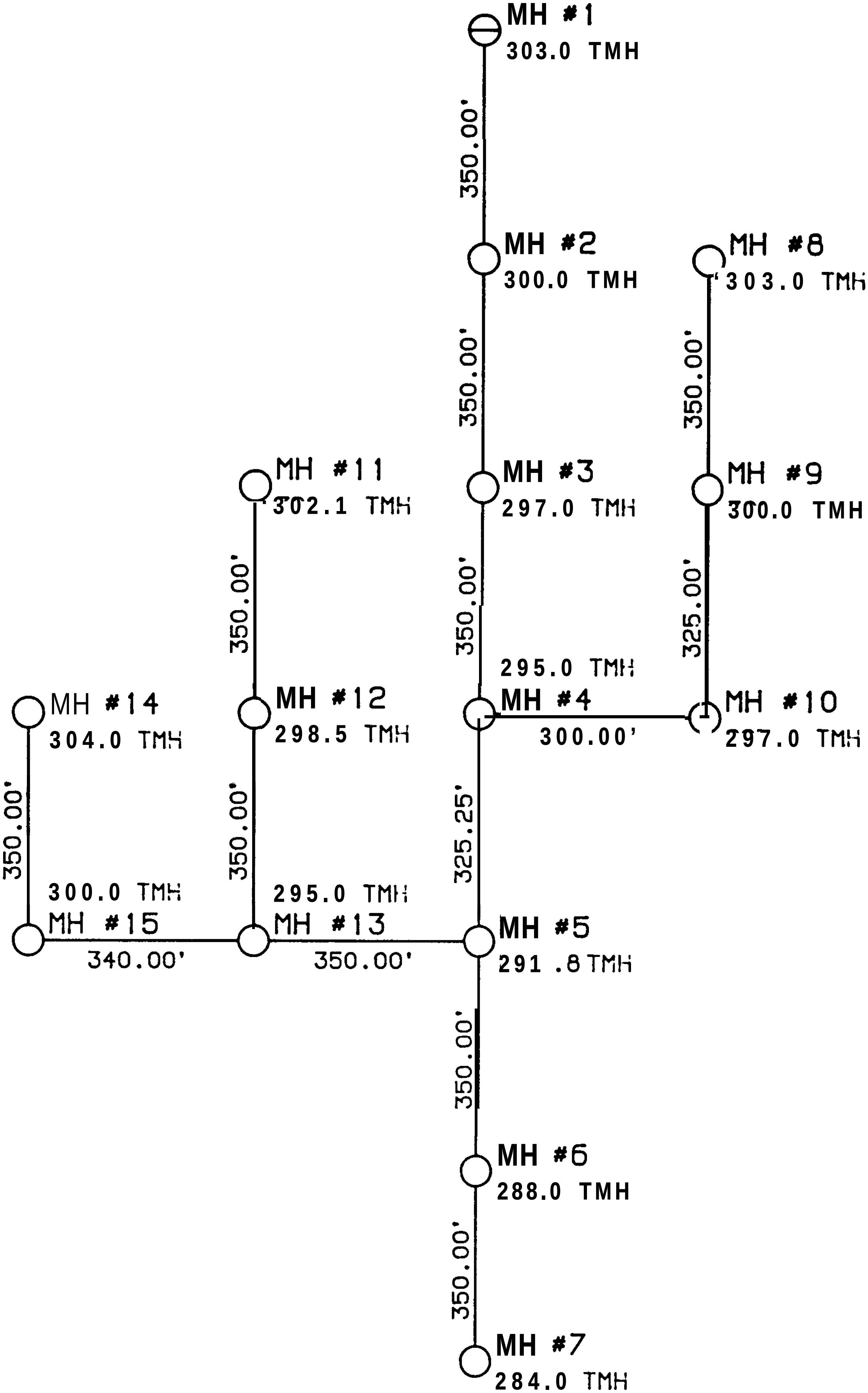
If Option #6 is selected for the ***beginning*** manhole, you will be asked for the manhole number, the flow coming into the system, and the pipe outlet invert elevation at that manhole.

If Option #1 is selected, you will be asked for:

> The manhole numbers and top-of-manhole elevation for the first two manholes (MH #1 and MH #2) and the distance between them. If you are using the station option, rather than ***distance between manholes, you*** will be asked for the station of each manhole.
>
> You will then be asked to enter the ***Development Type,*** and the ***area*** or ***capita,*** for this line. Depending upon the infiltration criteria for the entered ***development type, you*** may or may not be asked for infiltration data for this line.
>
> You will then be asked for the ***number of pipes to be*** *used* ***along this reach.*** You can use parallel pipes of the same size, if desired. Normally you will use one, which is the ***default*** value displayed.
>
> If you entered more than one manhole width (inside diameter), you will now be asked to enter the desired manhole for each manhole.

The entered and calculated data for this line segment will now be displayed. If it is O.K., enter [1] to continue. You also have the option to reenter data for this line, to override the calculated data for this line, or to backup in the program and change any of the design criteria.

The CRT will then display the Sewer *Flow* ***Process Options*** Menu, as shown on page ***6.***



EXAMPLE SEWER SYSTEM

FIGURE 11-1

```
1 - BEGINNING of sewer line (no external flow at beginning
    point of line)
2 - ADDITION to line and flow to next manhole
3 - PIPE flow to next manhole with no flow addition
4 - SIPHON (Inverted siphon or depressed sewer)
5 - LIFT or PUMP including FORCE MAIN
6 - USER SPECIFIED entry/override of flow in a line
7 - JUNCTION at point in current sewer line
```

After the data has been entered for the initial area (beginning manhole), you would select menu item #2 (ADDITION to line) to extend the pipe to the next manhole, with sewage flow added along the reach. You will then be asked for data for MH #2 and MH #3, similar to that asked for from MH #1 to MH #2 described above. Previously entered data for MH #2 will be displayed as default values.

As data for each line segment is entered, the program calculates the pipe size and slope required to maintain the minimum velocity and depth/ diameter for the flow through the reach. The added area or capita criteria is used to add to the previous flow rate.

This process is repeated until data has been entered to MH #4, which is a secondary junction manhole. After data has been entered from MH #3 to MH #4, you must select menu item #7 to indicate that MH #4 is a secondary junction (another line joins at MH #4). The CRT will then ask for the total number of secondary lines coming into this manhole. Do not count the outflow line (ie, from MH #4 to MH #5). In the case of MH #4, the number of secondary lines is two (MH #1 to MH #4, and MH #8 to MH #4). The CRT will then ask for the number of this secondary line (the one you have just entered data for), and since it was the first line entered, enter [1].

The CRT will then redisplay the abbreviated menu, with just Option #1 and Option #6. You must now enter data for the second secondary line coming into MH #4. All secondary lines into a secondary junction must be entered before moving beyond the junction (ie, entering data for MH#4 to MH #5) or before starting another line somewhere else in the system. In other words, you must enter the data from MH#8 to MI-I #9 to MH #10 to MH #4 before entering any other data.

Enter data for MH #8 to MH#9, to MH #10, and to MH #4. Then select menu item #7 to once again indicate that MH #4 is a secondary junction, and that the secondary line just entered was number two.

The CRT will now display all menu options, and you should select menu item #2 to add the line between MH #4 and MH #5. After this data is entered, select menu item #8 to indicate that MH #5 is a main line junction. It has two lines coming in, but since one of them contains a secondary junction (MH #13), it is a main junction, not a secondary junction. There are two lines coming in, and you have entered line number 1 (MH #4 to MH #5).

The CRT will then redisplay the abbreviated menu, with just Option #1 and Option #6. You

must now enter data for the second main line coming into MH #5, and you **must** begin at one of the ***beginning*** manholes (MH #11 or MH #14). For example, begin with MH #14 and enter data to MH #15 and then to MH #13. Then indicate that it is a secondary junction. Then start at MH #11 and enter data to MH #12 and then to MH #13. Again indicate that MH #13 is a secondary junction. Then add the line from MH #13 to MH #5. Indicate that MH #5 is a main line junction, and that the line you just entered was the second main line.

Then add the remaining lines to the outlet MH #7. After all data is entered, enter In] to exit back to the primary sewer system menu to create the report printout.

Junctions:

When reaching a point where two **or more incoming pipe lines join,** the *JUNCTION* option must be used.

A **main line** junction must be used if any of the additional incoming lines have one or more secondary junctions upstream. You must complete all lines into a main line junction before continuing past that junction, or before starting another line that does not eventually join at that junction.

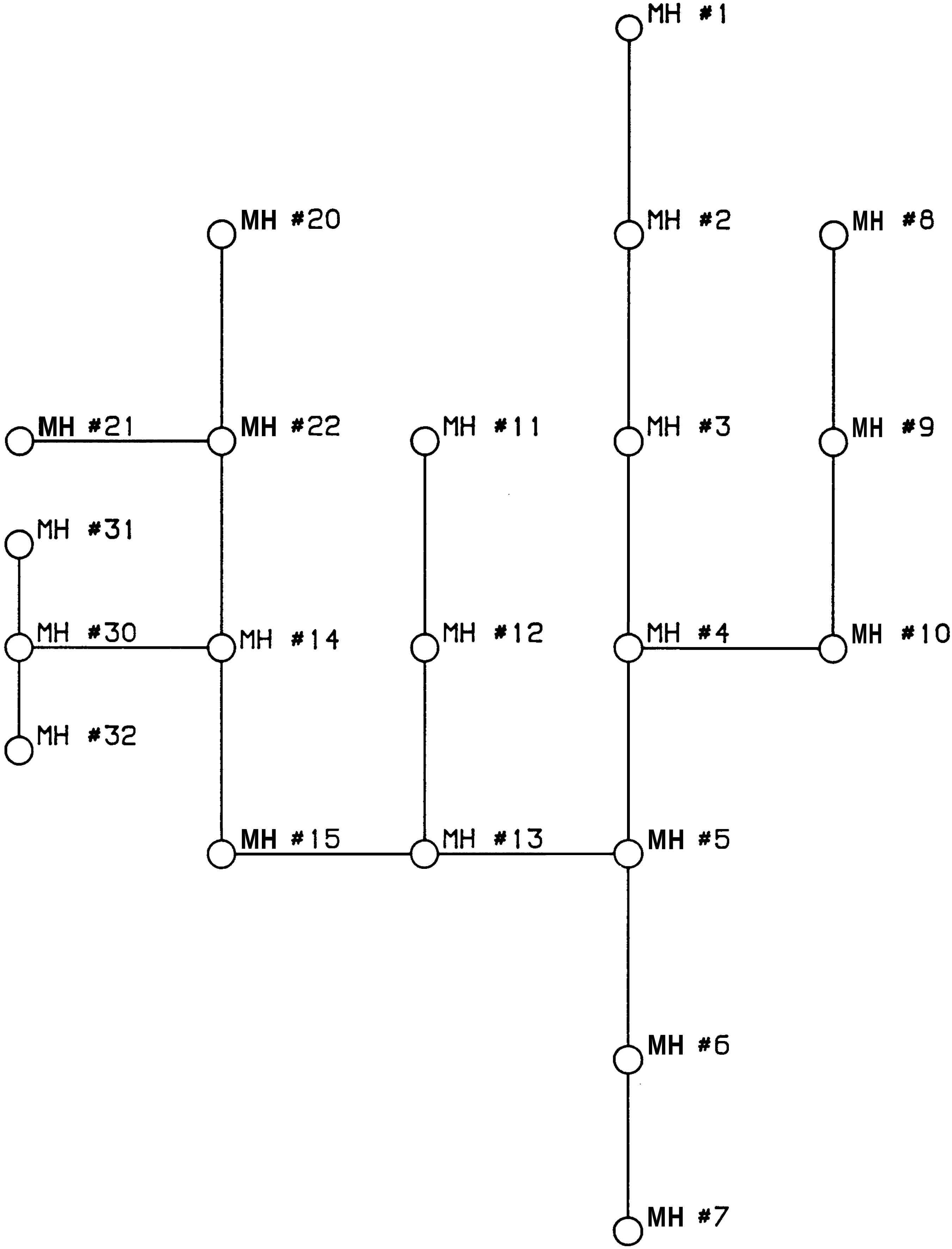
A **secondary junction** must be used when two or more incoming pipes join at a manhole, when the junction does not qualify as a ***main line junction.*** A **secondary junction cannot be used** if any of the additional incoming lines have any junctions upstream. In this situation, a ***main line junction*** must be used. You must complete all secondary lines into a secondary junction before continuing past that junction, or before starting another line that does not eventually join at that junction.

Note the term ***additional lines. The*** line originally used to get to the junction manhole does not count. For example, in Figure 11-1 MH #5 is a main junction because the additional line coming from MH #13 contains a secondary junction. The fact that MH #4 was a secondary junction does not matter, since it was upstream in the line used to get to MH #5.

The program can only handle **ONE main and/or ONE secondary junction at a time. All lines into any junction must be completed before you can move on. The example** sewer system shown in Figure 11-1 is a relatively simple system, and consequently there are four possible ***beginning*** manholes (#1, #8, #11, or #14). Regardless of which manhole (#1, #8, #11, **or** #14) ***you*** begin data entry with, each junction can be completed before you move on.

In a more complex sewer system, such as shown in Figure 11-2, you will need to analyze the system a bit more carefully to select the proper ***beginning*** manhole. Using Figure 11-2, if you begin at MH #1 and proceed down to MH #5, MH #5 is a main line junction since the line coming from MH #13 contains one or more junctions upstream from MH #5. However, that additional line contains another main line junction at MH #14. Consequently, you would be stymied (stuck), because you cannot process another main junction until you have completed all lines into the current main line junction.

Thus, in Figure 11-2, the only ***beginning*** manholes that can be used are MH #20, MH #21, MH #31, or MH #32. MH #22 and MH #30 will be secondary junctions, and MH #14 will



JUNCTION ANALYSIS
FIGURE 11-2

be a **main line** junction. If you begin with MH #20 or MH #21, the ***additional incoming line*** from MH #30 to MH #14 will contain a secondary junction. Or if you begin with MH #31 or MH #32, the ***additional incoming line*** from MH #22 to MH #14 will contain a secondary junction. So, in either case, the ***additional incoming line*** will contain a secondary junction, and thus MH #14 must be a main line junction. Since it will be completed before you reach MH #5, you will not be stymied. MH #5 will still be a main line junction, since the ***additional incoming line*** from MH #4 contains a secondary junction. After reaching MH #5 from MH #13, you would then select either MH #1 or MH #8 to begin entering data for the ***additional incoming line*** to MH #5.

When arriving at a junction, you must tell the program the total number of incoming pipes that are joining, and the number of the first line just entered (number the lines starting with 1 in sequence up to a maximum of 5). Until the junction is complete, you must start each additional incoming line to the junction using either menu Option #1 or menu Option #6.

After the last line has been joined at the junction, the current flow rate is calculated by summing all of the incoming pipe flows. Then the outflow pipe from the junction may continue routing down to the next junction point. When reaching the next confluence point, the junction sequence is started again.

Note: If your sewer system is a very complex network, with numerous secondary and main line junctions, it may be easier to break the network into several ***studies. The*** results (flow) of one study can be added to the beginning of the ***next study*** by using the User *Specified* ***Entry/Override*** option (menu item #6 in the Sewer Flow Process Options Menu).

Other Sewer Flow Process Options:

User Specified Entry/Override: In the ***Design Mode, you*** may override the lower manhole invert elevations that are calculated for each pipe. The feature is useful if obstacles are encountered that prohibit using the program calculated slopes in a particular sewer line segment. You may also override the calculated flow in any sewer segment, if desired, or if necessary to reflect flow conditions that cannot be calculated by the program.

No Added Flow: Should the system have a segment where no area flow is added, the ***Pipe*** *Flow* option should be selected to calculate the pipe size, slopes and elevations without adding flow to the pipe.

Siphon: If sewage flow is required through a ground depression, or under any obstruction which requires the pipe to flow full, the ***Siphon*** option should be used. This option uses pressure flow equations to calculate the head loss through the siphon and the elevations required to maintain pipe flow through the siphon. No added area flow is included in this option. You must enter the size of pipe(s), however you are not limited to those pipe sizes entered in the General ***Data.*** Select a pipe size that will give a velocity high enough to self cleanse the bottom of the siphon. You must also enter the actual length of siphon pipe (will normally be greater than horizontal distance between manholes), and the minor loss *K* factor. Typical *K* factors can be found in our ***Pipe Network Analysis*** operators manual, as well as in many hydraulics textbooks. Note that the actual configuration of the siphon, ie depth, bends, etc. is not entered; however, the actual configuration must be known to determine the length of the siphon and the *K* factor that are entered.

Lift Pump/Force Maim This option is included for locations where the sewage must be raised or forced to flow uphill. This option will calculate the flow requirements and 100% efficiency pump power requirements to achieve the flow rate and lift required. The option calculates the head loss through the pipe(s) or ***Force Main,*** the lift elevation required, and uses an energy equation to calculate the 100% efficiency power required.

7.2 - Check Mode

The ***Check Mode*** of operation evaluates an existing sewer network and determines pipe capacities (depth of flow) throughout the existing system. This mode can also be used to change or modify a ***Design Mode*** data file to fit particular dimensions or requirements.

Your sewer system layout should be identical to the layout sketch described in paragraph ***7.1*** above, except you will also need existing invert elevations for the centerline of each manhole.

The General Data entries are identical to those described in paragraph 7.1 above.

Data entries for the manholes, junctions, etc. is similar to those described for the ***Design Mode*** in paragraph 7.1 above. There are several additional entries required, however. You will be asked for the pipe size and the invert elevation for the centerline of each manhole. The calculated outlet invert elevation for the upper manhole and the inlet invert elevation for the lower manhole will be displayed. If they are correct, default those entries, or enter the correct outlet or inlet invert elevation.

Junctions are defined and handled exactly as described in paragraph 7.1 above.

The program program uses the sewage and infiltration criteria and other entered data to calculate the flow and depth of flow throughout the system.

7.3 - Create Report Files

After all data has been entered and you have returned to the *Sewer* ***Network Program Options*** Menu, select menu item ***#2*** to create a ***Standard Detailed Report*** or menu item ***#3*** to create a *Summary* ***Form Report.***

The standard detailed report includes the same data that appeared on the CRT when building the file. This report may either be sent to the printer, or to an output file for later *VUEing* or printing, or to the screen for review.

The other report is in a summary format which requires a printer width of 132 characters. This report may also be saved as a file or sent to the printer.

7.4 - Edit Data File

You may make changes to entered data, and/or insert or delete options previously used when creating your data file. Enter [4] from the Sewer ***Network Program Options*** Menu to edit your data.

Then select the ***Operation Mode*** used to create the original data, either Design or ***Check. You*** then have the option to edit the General ***Data*** entries, if necessary, or to skip those items and go directly to data entered for the sewer lines. If the ***General Data*** entries are edited, previous entries will be displayed as ***defaults.*** Default [RTN] those that are correct, and enter new data for those you want to change.

The CRT will then display each sewer segment, with the upper and lower manhole numbers for each segment. Enter the line number of the sewer segment that you want to edit, and the CRT will display four edit options. You can change data for the entered sewer segment; or you can add a new segment above or below the entered segment; or you can delete the entered sewer segment entirely.

If you need to change data, the old data is shown in reduced intensity, and if it is O.K., simply default [RTN]. If a change is required, enter the new data and the old information will be replaced.

7.5 - Add to Data File

If you need to exit the sewer network program before all sewer segments data is entered, or if you want to add additional segments at some later date, you may reaccess the program and select the ***ADD To Data File*** option. When this option is selected, the program will process the entire existing file to calculate the flow rates up to the end of the current file.

You can then begin adding new data to the file. After all data has been entered, enter In] to return to the main menu.

7.6 - Help File

There is a detailed ***Help File*** that can be accessed from the Sewer ***Network Program Options*** Menu, menu item #7. The ***Help File*** contains examples of sewage and infiltration constants used in various areas, as well as general information on sanitary sewer system design and design criteria, and is recommended reading.

7.7 - Profile Data Files

Proposed sewer profile data files can be generated, if you enter manhole numbers as explained earlier in this section. They are generated by accessing the ***Develop/Edit Utilities Profiles*** program described in Section 3, Chapter 3. When in that program level and when creating a sewer profile, you will be given the option of using the output data file from this sewer design program to develop the proposed sewer profile. Your entries will be the manhole numbers, in sequence from lower to higher station, that will make up the particular profile being developed. For example, you might develop one profile consisting of MH #1 through MH #7, another from MH #4 to MH #8, etc. See Section 3, Chapter 3, for detailed instructions.

(This page intentionally left blank)

Appendix A

PC Loading Instructions

The furnished diskettes contain the executable code for the programs that you have been licensed to use **on a single PC.**

To load these programs on your PC, follow the distribution letter instructions, or:

1. The computer (PC) must have at least 550k of memory; use MS-DOS Version 4.0 or later or Windows 95; and have a hard disk setup.

2. The CONFIG.SYS file must have the statement DEVICE = **C:\(path)\ANSI.SYS**. The path is normally the DOS directory, or, if you are using Windows 95 software, the path is **WINDOWS\COMMAND**.

3. **Also,** the CONFIG.SYS file should have the statements FILES=20 and BUFFERS=20 or equivalent.

4. The AUTOEXEC.BAT file should have the statement PRINT to install the DOS print module.

5. All diskettes should be copied into a single directory, such as **C:\CIVILD,** and the AUTOEXEC.BAT file should contain a path statement to that directory of the programs are to be utilized in a different directory.

6. To run the programs, type the command **CIVILD**, or, type the name of the individual program, and then follow the instructions in the program and the associated Sections of the operators manual. Some programs also have a documentation file (**.DOC**), and these files should be printed on your printer for added instructions.

(This page intentionally left blank)