US Army Corps of Engineers

AD-A224 387

(2)

COMPUTER-AIDED STRUCTURAL ENGINEERING (CASE) PROJECT

TECHNICAL REPORT ITL-90-3

INVESTIGATION AND DESIGN OF U-FRAME STRUCTURES USING PROGRAM CUFRBC

VOLUME A

PROGRAM CRITERIA AND DOCUMENTATION

by

Clifford O. Hays, Jr.

1910 SW 43rd Avenue
Gainesville, Florida 32611

May 1990

Final Report

Approved For Public Release; Distribution Unlimited

Prepared for DEPARTMENT OF THE ARMY
US Army Corps of Engineers
Washington, DC 20314-1000

Monitored by Information Technology Laboratory
US Army Engineer Waterways Experiment Station
3909 Halls Ferry Road, Vicksburg, Mississippi 39180-6199

2

Unclassified
SECURITY CLASSIFICATION OF THIS PAGE

REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-0188

1a. REPORT SECURITY CLASSIFICATION: Unclassified	1b. RESTRICTIVE MARKINGS
2a. SECURITY CLASSIFICATION AUTHORITY	3. DISTRIBUTION/AVAILABILITY OF REPORT: Approved for public release; distribution unlimited
2b. DECLASSIFICATION/DOWNGRADING SCHEDULE	
4. PERFORMING ORGANIZATION REPORT NUMBER(S): Technical Report ITL-90-3	5. MONITORING ORGANIZATION REPORT NUMBER(S)
6a. NAME OF PERFORMING ORGANIZATION: See reverse. / 6b. OFFICE SYMBOL (If applicable)	7a. NAME OF MONITORING ORGANIZATION: USAWES Information Technology Laboratory
6c. ADDRESS (City, State, and ZIP Code)	7b. ADDRESS (City, State, and ZIP Code): 3909 Halls Ferry Road, Vicksburg, MS 39180-6199
8a. NAME OF FUNDING/SPONSORING ORGANIZATION: US Army Corps of Engineers / 8b. OFFICE SYMBOL (If applicable)	9. PROCUREMENT INSTRUMENT IDENTIFICATION NUMBER
8c. ADDRESS (City, State, and ZIP Code): Washington, DC 20314-1000	10. SOURCE OF FUNDING NUMBERS: PROGRAM ELEMENT NO. / PROJECT NO. / TASK NO. / WORK UNIT ACCESSION NO.

interactive

11. TITLE (Include Security Classification)
Investigation and Design of U-Frame Structures Using Program CUFRBC;
Volume A: Program Criteria and Documentation

12. PERSONAL AUTHOR(S)
Hays, Jr., Clifford O.

13a. TYPE OF REPORT	13b. TIME COVERED	14. DATE OF REPORT (Year, Month, Day)	15. PAGE COUNT
Final report	FROM ____ TO ____	May 1990	136

16. SUPPLEMENTARY NOTATION
Available from National Technical Information Service, 5285 Port Royal Road, Springfield, VA 22161.

17. COSATI CODES			18. SUBJECT TERMS (Continue on reverse if necessary and identify by block number)
FIELD	GROUP	SUB-GROUP	Basins; Channels; Drainage; Hydraulic structure; Reinforced concrete; Structural design; U-frame

19. ABSTRACT (Continue on reverse if necessary and identify by block number)

The computer program CUFRBC can be used to investigate or design basins or channels for a variety of load conditions based on a two-dimensional frame analysis of a 1-ft slice of the U-frame.

The soil loading on the walls may be obtained by empirical coefficients, active or passive wedge analyses with corrections for at-rest conditions, or inputing force-deformation curves for the walls. Hydraulic loads are automatically computed from water elevations and drain data. Foundation reaction pressures may be computed using a simple equilibrium approach or a Winkler spring on elastic foundation model.

Design may be by allowable stress or strength design procedures, using American Concrete Institute or Corps criteria. Output includes member pressures, shears, moments, and stress or strength results at discrete points. Graphical output is available.

20. DISTRIBUTION/AVAILABILITY OF ABSTRACT: ☒ UNCLASSIFIED/UNLIMITED ☐ SAME AS RPT ☐ DTIC USERS	21. ABSTRACT SECURITY CLASSIFICATION: Unclassified
22a. NAME OF RESPONSIBLE INDIVIDUAL	22b. TELEPHONE (Include Area Code) / 22c. OFFICE SYMBOL

DD Form 1473, JUN 86 — Previous editions are obsolete. — SECURITY CLASSIFICATION OF THIS PAGE: Unclassified

Keywords: Fortran, U-Frame Basin and Channel Structures, CASE (Computer Aided Structural Engineering), Army Corps of Engineers. (KR)

90 07 31 028

6a. NAME OF PERFORMING ORGANIZATION (Continued).

Basins and Channels Task Group Computer-Aided Structural Engineering Project

ELECTRONIC COMPUTER PROGRAM ABSTRACT

TITLE OF PROGRAM Investigation and Design of U-Frame Structures Using Program CUFRBC			PROGRAM NO.
PREPARING AGENCY			
AUTHOR(S) Clifford O. Hays, Jr.	DATE PROGRAM COMPLETED September 1989	STATUS OF PROGRAM — PHASE Final	STAGE

A. PURPOSE OF PROGRAM

The computer program CUFRBC can be used to investigate or design basins or channels for a variety of load conditions based on a two-dimensional frame analysis of a 1-ft slice of the U-frame. Effects of drains and anchors may be included, and the program offers a variety of options concerning the computation of soil pressures. Thus, the program has sufficient versatility to suffice for preliminary designs, final designs, or in-depth investigations.

B. PROGRAM SPECIFICATIONS

Time-sharing FORTRAN Program.

C. METHODS The soil loading on the walls may be obtained by empirical coefficients, active or passive wedge analyses with corrections for at-rest conditions, or inputing force-deformation curves for the walls. Hydraulic loads are automatically computed from water elevations and drain data. Foundation reaction pressures may be computed using a simple equilibrium approach or a Winkler spring on elastic foundation model. For all loadings, a frame analysis is made to generate internal forces and moments at discrete points along the members. Design may be by allowable stress or strength design procedures, using American Concrete Institute or Corps criteria.

D. EQUIPMENT DETAILS

A data entry terminal is required to operate the program in the time-sharing mode. A Techtronix graphics device or emulator is required for obtaining graphical output.

E. INPUT-OUTPUT

Data can be input interactively with the aid of an on-line editor or from a prepared data file with or without line numbers. Output includes member pressures, shears, moments, and stress or strength results at discrete points. Numerical output can be displayed at the terminal or directed to an output file. Graphical output is available using a companion program CUFRMP and the Corps graphics package GCS2D.

F. ADDITIONAL REMARKS

PROGRAM INFORMATION

Description of Program

CUFRBC, called X0095 in the Conversationally Oriented Real-Time Programming System (CORPS) library, can be used to investigate or design basins or channels for a variety of load conditions based on a two-dimensional frame analysis of a 1 ft. slice of the U-frame. Effects of drains and anchors may be included, and the program offers a variety of options concerning the computation of soil pressures. Thus, the program has sufficient versatility to suffice for preliminary designs, final designs, or in-depth investigations. Graphical output is available using a companion program, X0096 (CUFRMP).

Coding and Data Format

CUFRBC is written in FORTRAN and was developed on the Power Computing Company Cyber 865. It will be available in the future on the following systems:

a. WES Honeywell DPS/8

b. Local District Harris 500 Series.

c. Micro Computer IBM PC/XT/AT compatibles.

d. Intergraph workstations.

How to Use CUFRBC

A short description of how to access the program on each of the systems, when the program is available, is provided. It is assumed that the user knows how to sign on the appropriate system before trying to use CUFRBC. In the example initiation of execution commands that follow, all user responses are underlined, and each should be followed by a carriage return.

WES Honeywell System

The user signs on the system and issues the run command.

<u>FRN WESLIB/CORPS/X0095,R</u>

to initiate execution of the program. The program is then executed as described in this user's guide. The data file should be prepared prior to issuing the FRN command. An example initiation of execution is as follows, assuming a data file had previously been prepared:

COEWES HIS TIMESHARING ON 05/10/90 AT 11.612 CHANNEL 2426 TS2

USER ID --<u>ROKACLA</u>
PASSWORD--
<u>XXXXXXXX</u>
#USERS=016 SS=0247K %MEM-USED=046 000-WAIT-000K

*<u>FRN WESLIB/CORPS/X0095,R</u>

Power Computing Company
Computer System

The log-on procedure is followed by a call to the CORPS procedure file

OLD,CORPS/UN=CECELB

to access the CORPS library. The file name of the program is used in the command

BEGIN,,CORPS,X0095

to initiate execution of the program. An example is:

CONNECTED TO (20) 5-2
90/05/10. 11.34.45. AA1D8HA
SN1048 POWER COMPUTING COMPANY NOS1.4-531-795-A
FAMILY: KOE
USER NAME: CEROF8
PASSWORD
XXXXXXXX
TERMINAL: 6, NAMIAF
RECOVER/ CHARGE: CHARGE,CEROEGC,CEROF8
$CHARGE,CEROEGC,CEROF8.
/OLD,CORPS/UN=CECELB
/BEGIN,,CORPS,X0095

Harris System

The user signs on the system and issues the run command

*CORPS,X0095

to initiate execution of the program.

An example is:

"ACOE-WES(H500 V7.1.0)"
ENTER SIGN-ON
1KABC ROKABC
ENTER PASSWORD
XXXXXXXX

** GOOD MORNING CORPS-LIB, IT'S 10 MAY 90 11:34:51
WES HARRIS 500 FOR SYSTEM INFORMATION - ENTER *NEWS
*CORPS,X0095

How to Use CUFRMP

Commands for execution of the companion program CUFRMP are similar. The user replaces the program number X0095 in the above examples with X0096.

How to Use CORPS

The CORPS system contains many other useful programs which may be catalogued from CORPS by use of the LIST command. The execute command for CORPS on the WES system is:

```
*FRN WESLIB/CORPS/CORPS,R
ENTER COMMAND (HELP, LIST, BRIEF, EXECUTE OR STOP)
*?LIST
```

On the Power computing Company computer system, the commands are:

```
/OLD,CORPS/UN=CECELB
/BEGIN,,CORPS,CORPS
ENTER COMMAND (HELP, LIST, BRIEF, EXECUTE OR STOP)
*?LIST
```

On the Harris computer system, the commands are:

```
*CORPS
ENTER COMMAND(HELP,LIST,BRIEF,EXECUTE OR STOP)
*?
```

PREFACE

This report, Volume A - "Program Criteria and Documentation," documents and gives the development criteria for an interactive computer program CUFRBC, a program for interactive investigation and design of U-Frame Basin and Channel structures. The program was developed and the report written using funds provided to the US Army Engineer Waterways Experiment Station (WES), Vicksburg, Mississippi, by the Civil Works Research and Development Program of the US Army Corps of Engineers (USACE), US Army, under the Structural Engineering Research Program Work Unit entitled "Computer-Aided Structural Engineering (CASE) Project."

Volume B, "User's Guide for Basins," gives instructions for routine use of the program for basin structures. Volume C, "User's Guide for Channels," gives instructions for routine use of the program for channel structures.

The program was prepared with criteria developed by the Basins and Channels Task Group of the CASE Project. Members of this group during program development were:

Mr. Byron Bircher, CEMRK-ED-D, Chairman, U-Frame Structures Task Group
Mr. George Henson, CESWT-EC-DT, Chairman, U-Frame Basins and Channels Sub Group
Mr. Frank Coppinger, CENAD-EN-TF
Mr. Edwin Aling, Soil Conservation Service (formerly)
Mr. Donald Dressler, CEEC-ED-D
Mr. Clifford Ford, CESPL-ED-DB
Mr. Lucian Guthrie, CEEC-ED-D
Mr. Bill James, CESWD-ED-TS (formerly)
Mr. Ivar Paavola, CEEC-ED-D (formerly)
Mr. Mike Pace, CEWES-IM-DS
Mr. William Price, CEWES-IM-DA
Mr. Scott Snover, Soil Conservation Service (formerly)
Mr. Tom Wright, CEMRK-ED-DT

The computer program and this document were written by Dr. Clifford O. Hays, Jr., P.E., Gainesville, Florida, under contracts with WES. Mr. William Price, Information Technology Laboratory (ITL), monitored the contract and coordinated the work. The work was done under the supervision of Dr. N. Radhakrishnan, Chief, ITL, and Mr. Paul K. Senter, ITL. Mr. Donald Dressler was the point of contact with USACE.

COL Larry B. Fulton, EN, is the Commander and Director of WES. Dr. Robert W. Whalin is the WES Technical Director.

CONTENTS

Page

CONVERSION FACTORS, NON-SI TO SI (METRIC) UNITS OF MEASUREMENT

Non-SI units of measurement used in this report can be converted to SI (metric) units as follows:

Multiply	By	To Obtain
feet	0.3048	metres
inches	2.54	centimetres
kips (force)	4.448222	kilonewtons
kips (force)-feet	1355.818	newtons-metres
kips (force) per square inch	6894.757	kilopascals
kips (force) per square foot	47.88026	kilopascals
pounds (force) per cubic foot	0.157087	kilonewtons per cubic metre
pounds (force) per square inch	6.894757	kilopascals
square inches	6.4516	square centimetres

INVESTIGATION AND DESIGN OF U-FRAME STRUCTURES USING PROGRAM CUFRBC

VOLUME A - PROGRAM CRITERIA AND DOCUMENTATION

PART I: INTRODUCTION

Description of Program

1. The computer program CUFRBC is a CASE program for interactive investigation and design of U-FRame Basin and Channel structures. The program offers a large number of options and features that allow the engineer to obtain results based on a number of different design assumptions. For examples, the program accommodates both basin or channel geometries, soil pressures may be computed by several options, and section strength may be assessed by elastic or strength procedures. However, the program is written such that users may prepare their input in the interactive mode without being concerned about input details that do not apply to the chosen assumptions.

2. The user of the program should be familiar with the various options offered in order to choose the program features that most closely model the behavior of his or her structure. Separate user's guides for basins and channels are provided in Volumes B and C of this report.

3. While the program operates in either investigation or design modes, the design mode is little more than a predetermined series of analyses and checks of design criteria. Thus, all features of the investigation mode are presented prior to introducing the design mode. It is essential that the user of the program be thoroughly familiar with the analysis procedures in order to properly apply the design features.

4. The U-frame is modeled as a framed planar s' cture using a frame analysis module, FRAME55, which is a modification of eneral nonlinear frame analysis program, FRAME54 (Hays 1971, 1982). The earlier program has quite general nonlinear analysis capabilities, allowing for nonlinear material, geometric and soil-structure modeling through soil force-deformation curves. However, only the nonlinear soil support features are utilized in FRAME55. Input requirements for FRAME55 are excessive for routine U-frame structures. In addition, output from FRAME55 is not in the form most useful to U-frame

designers. T. .: a preprocessor and postprocessor were written for FRAME55, and some minor modifications were made to the frame analysis program to create CUFRBC, a user friendly interactive program capable of quickly analyzing or designing a U-frame structure for a variety of different load and support conditions.

5. Graphical output of the results may be obtained. This output allows designers to quickly verify that their data were interpreted correctly by the program and to visualize the results. The ease with which the data files may be modified and the program rerun allows the designer to quickly study the effects of physical parameters that are not well defined. Thus, investigations and designs may be obtained for envelopes of parameters.

6. The input needed by the frame analysis module is generated by the CUFRBC program from a minimum of input of physical parameters defining the outline of the structure and the soil and water geometry. The U-frame structure may have from one to three bays and be quite general in its geometric configuration, as defined subsequently. Volumes B and C of this report are separate user's guides for basin and channel structures, respectively. Once the program user specifies whether his or her structure is a basin or a channel, all prompts in the program are specific to the user's type of structure. However, this document (Volume A) describes both basin and channel structures.

7. The frame model described subsequently in detail allows for a variety of loading and support options for the members of the frames. The members are the walls, portions of base slab between walls, and heels. The heels are the extensions of the base slab beyond the end walls.

8. The self-weight of the U-frame is automatically included in all load cases. Hydraulic loads on all the members are computed within the program from input of water elevations, locations of wall and base slab drains, and drain efficiencies. Earth pressure on the walls and top of heels may be computed using: (a) an empirical approach that uses simple effective lateral soil coefficients, (b) wedge solutions for active or passive loadings including surcharges, or (c) nonlinear lateral force-deformation curves. Any of these solutions may be done with a minimum of input describing the soil geometry and properties. Specialized loads such as those arising from wind or earthquake may be specified as general concentrated and distributed loads on the members. Equilibrium of lateral forces is provided by adjusting active

and passive solutions, including base shear effects or as part of the nonlinear force-deformation solution.

9. Resistive loading on the bottom of the base slab and heel may be computed to satisfy the vertical and rotational equilibrium from a simple statical empirical approach or using a compression only elastic spring foundation. Vertical tension only anchors may also be included.

10. Output from the frame analysis includes the distribution of various pressures against members and member forces (axial, shear, and moment) at discrete points along members. Sections may be investigated or designed by the traditional elastic theory (working stress design) or by the strength design procedure.

Purpose

11. The true nature of loading for basin and channel structures is quite complex. The best available models of such behavior are three-dimensional (3-D) finite element models which incorporate the construction sequence along with nonlinear soil-water-structure interaction effects. However, the use of such complex models is not generally accepted at present, since numerous U-frame structures have been successfully designed using a variety of essentially two-dimensional (2-D) frame models. In fact, the 2-D modeling procedures used by Corps designers have varied widely in their complexity. A major purpose in developing CUFRBC was to incorporate most of the 2-D modeling procedures such as the previously described loadings into one program.

12. A frame model is convenient for 2-D analysis of the U-frame structures, since the loadings can be made as simple or as complex as desired and the frame analysis will yield shears and moments that can be easily interpreted by designers. Thus, CUFFBC can be used as a design tool to economically duplicate most of the standard designs being presently executed. Perhaps more importantly, the program can be used to allow parameter studies to be made of the effects that both physical variables and various loading assumptions have on the design variables in typical U-frames. Finally, the program can be used to compare with finite element solutions and experimental data when available.

Disclaimer

13. This program has been developed using criteria supplied by the Basins and Channels Subgroup, U-FRAME Structures Task Group of the CASE Project. This volume describes the criteria and documents the assumptions on which the program is based. The program has been subjected to extensive testing by the author and members of the committee to ensure that it is reasonably error free and will generally provide reasonable analyses or designs for U-frame structures. However, no warranty of the correctness of the results for any particular structure is made or implied by the author. The user of the program is responsible to ensure that the assumptions inherent in the program are applicable to the structure chosen and that the numerical results are reasonable.

PART II: STRUCTURE GEOMETRY

14. The program allows for the investigation or design of basin and channel structures as subsequently described. The user of the program is warned against applying the program to other structures, which might superficially resemble the structures described herein but might be significantly different when loading or behavior is considered.

Basin Structure

15. Basins are typically used in outlet works, stilling basins, and approach spillways. Their criteria follow EM 1110-2-2400, "Structural Design of Spillways and Outlet Works" (Headquarters, Department of the Army 1961a). The program considers basins with from one to three bays as shown in Figures 1, 2, and 3. These figures show the geometric outlines and define the input variables further described in the "User's Guide for Basins" (Volume B of this report).

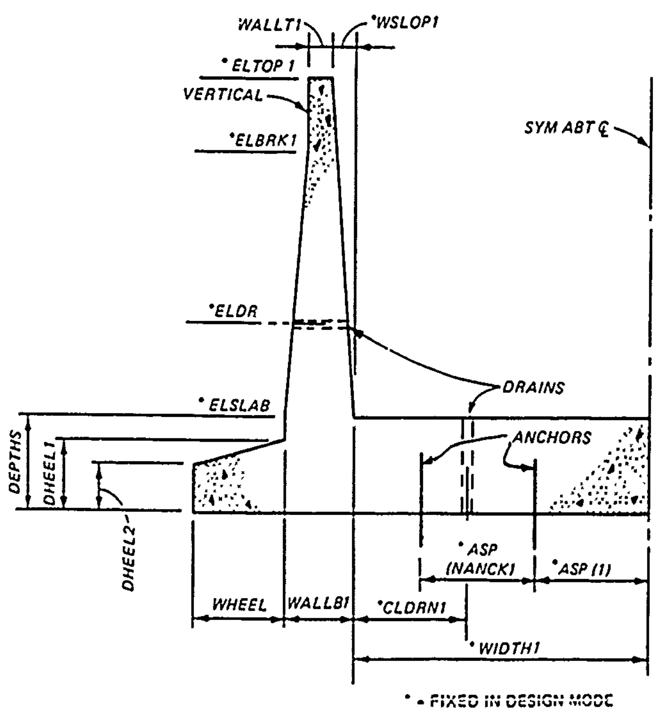


Figure 1. Single-basin structure

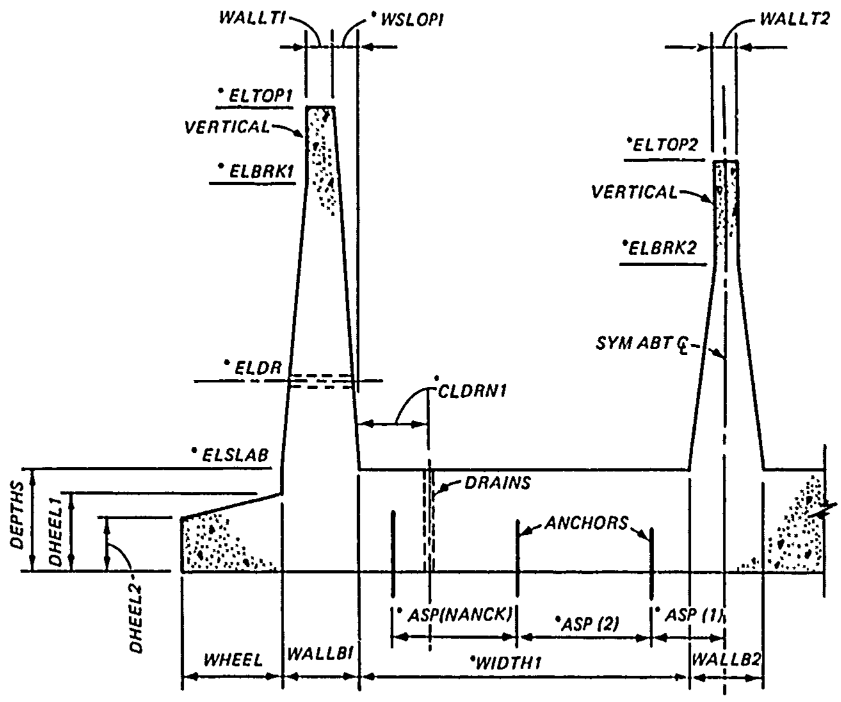


Figure 2. Double-basin structure

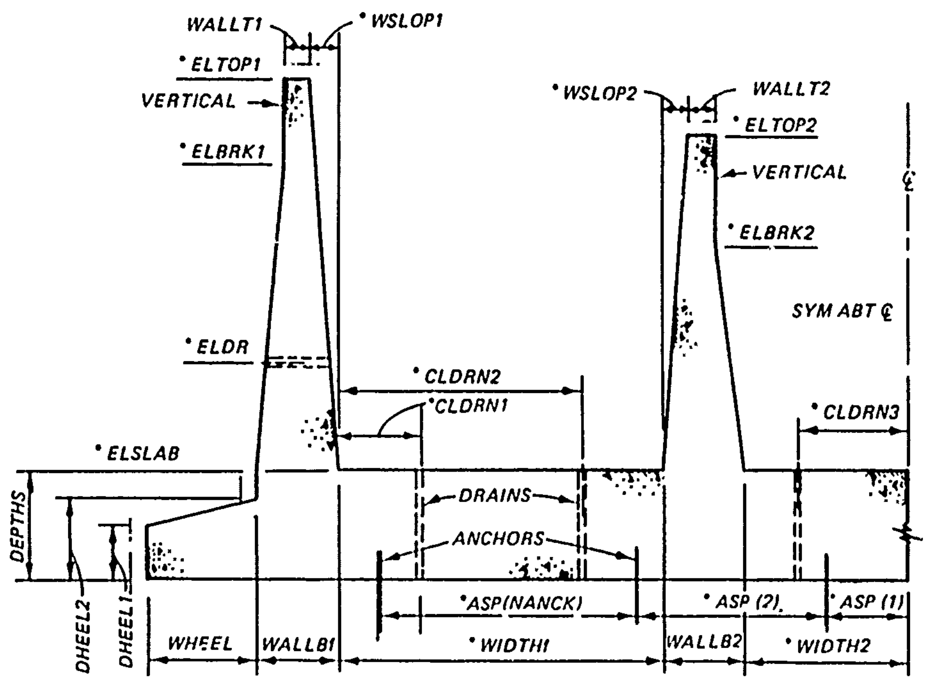


Figure 3. Triple-basin structure

16. The input values define the cross section for the investigation mode. However, in the design mode the input values define the initial cross section. Input variables shown with an asterisk are kept constant in the design mode. The cross-section variables not shown with an asterisk are incremented as necessary for the final design. In addition, the slope on the top face of the heel is kept constant during the design iterations. Details of the design procedure are given later in this report.

17. All three cross sections are assumed symmetrical, as is the case for almost all basin structures. Thus, the amount of input is reduced considerably. However, as discussed later, unsymmetrical loading and reinforcing are permitted in the investigation mode.

18. The variables describing the locations of drains and anchors are shown in the figures defining the geometric outlines of the basin. However, the use of these variables is discussed in subsequent sections.

19. Input and output for the basin are keyed to the members as defined in Figure 4. The details of the frame model are discussed subsequently. However, it is important to note that the frame analysis considers a frame

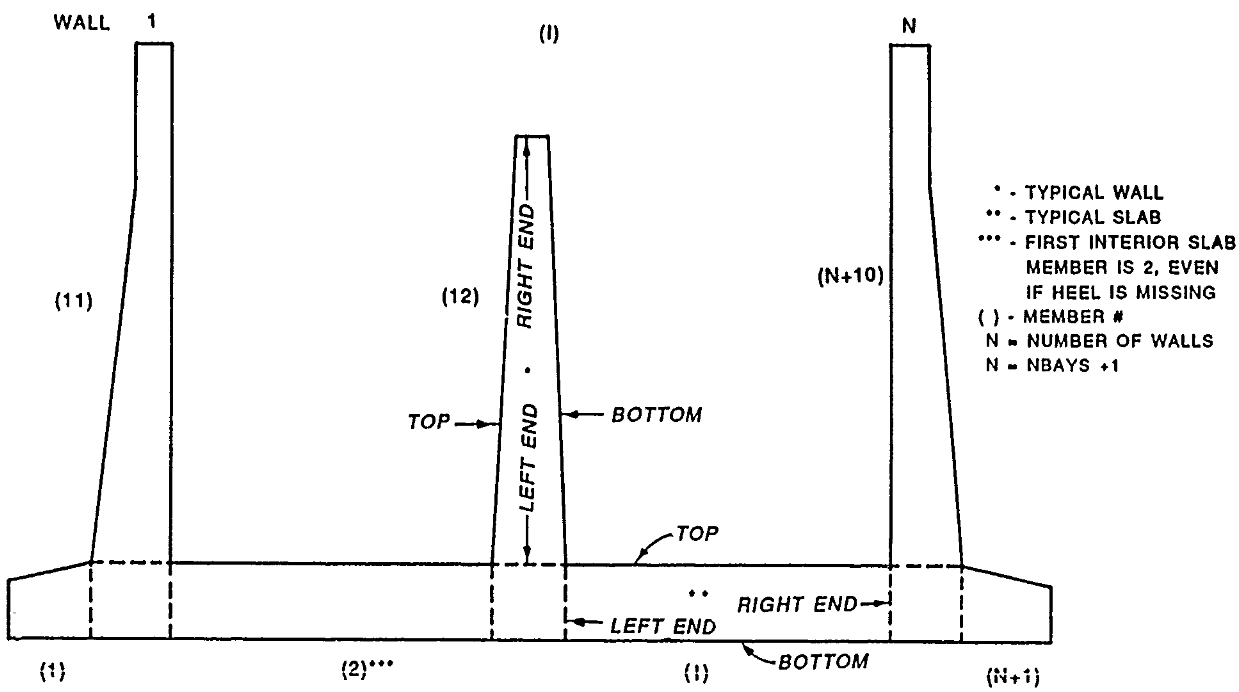


Figure 4. Basin geometry model

of relatively flexible vertical and horizontal members connected at essentially rigid joints of finite size. The rigid joints are shown within dashed lines in the figure. Base slab members including heels are numbered from left to right from 1 to N + 1, where N is the number of walls. Heels may be omitted; however, if they are omitted, the first actual slab member will still be referred to as member 2. The number of bays is NBAYS and

$$N = NBAYS + 1$$

The leftmost wall is numbered 11, and then the remaining walls increase in number from left to right as shown in the figure. Input of reinforcing and special loads and the all member output are keyed to these member numbers and the "left-right" - "top-bottom" orientation of the members as shown in the figure. Distances along the member are always specified from the "left" end of the member.

20. Reinforcement details for the investigation mode are shown in Figure 5. Sections may be reviewed by elastic or strength procedures at up to five points per member. The locations of the review points are specified from the "left" ends of the members as shown in the figure. Up to three layers of reinforcing may be specified for the "top" and "bottom" of a member. As many of the members as desired may be reviewed; NMINV is the total number of members being reviewed. Thus, if all members of a single basin structure with heels were reviewed, NMINV would be five (two walls, two heels, and the center slab).

21. It should be noted that the "top" layers of steel are not effective in resisting tension on the "bottom" side of the member. Thus, the user should ensure that steel is located in the proper face for all load conditions. Details on the calculations of elastic stresses and strength design procedures are discussed subsequently.

22. NTOPL and NBOTL are the number of layers in the "top" and "bottom" of the section, respectively. Layers are numbered from the exterior of section to the interior as shown in the figure. The steel within the layers may be specified by two different bar options. For 'REOPT' = "BAR," the steel within each layer is specified by the bar size (number of nominal

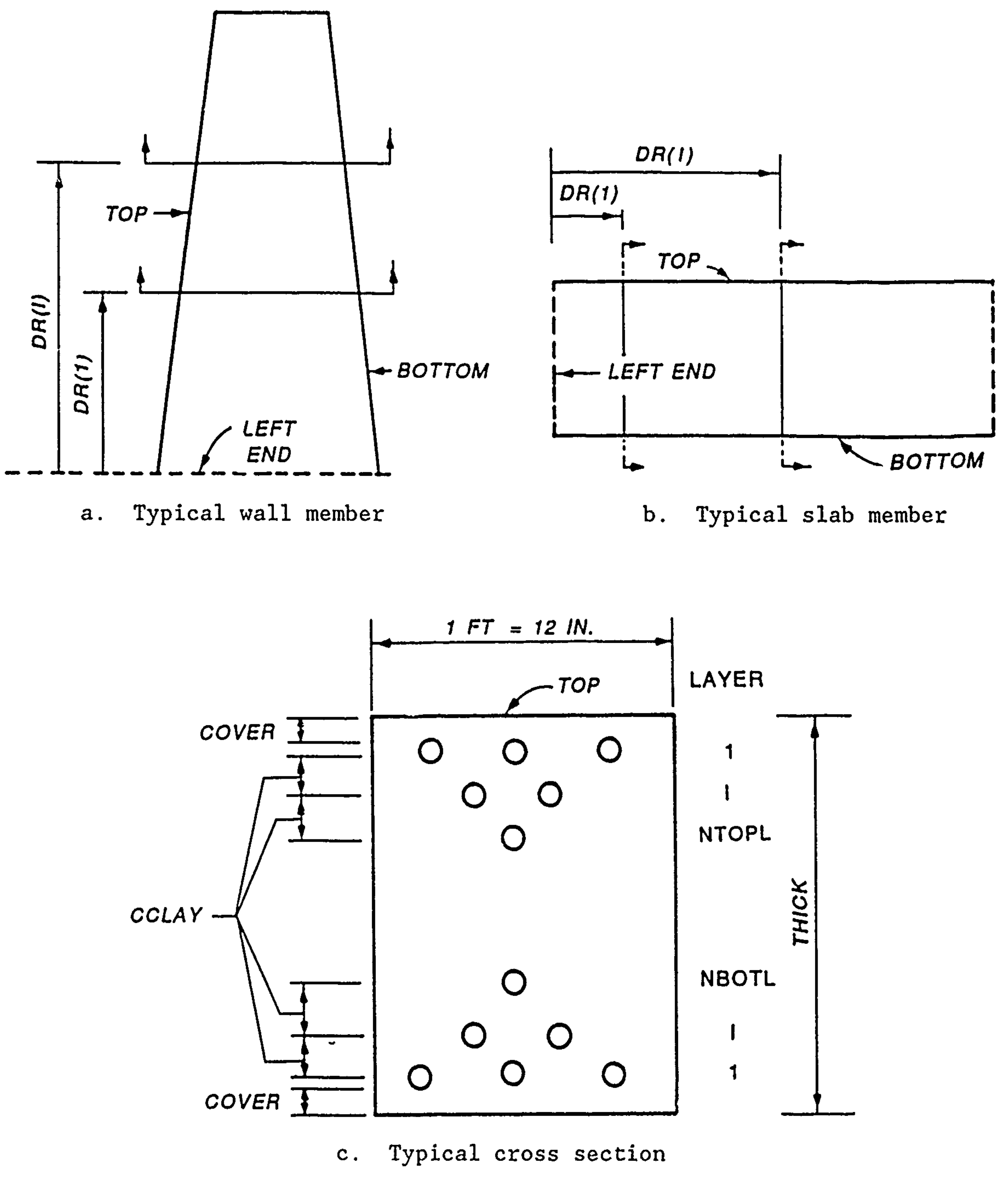


Figure 5. Description of reinforcement/analysis option

one-eighth-in.* increments in diameter) and the spacing in inches within the layer. For 'REOPT' = "ARE," the steel is specified by giving the area in square inches per foot of the steel in each layer and the nominal diameter of the steel in the outer layer. This nominal diameter is only used in computing the location of the centroid of the outer steel layer.

23. The variable COVER is the clear cover from the outer edge to the first steel layer and is specified for four different conditions as defined in the "User's Guide for Basins" (Volume B). The center-to-center distance between steel layers, CCLAY, is constant at all locations.

Channel Structure

24. Channels are typically used in floodways. Their criteria generally follow, but are not limited to, EM 1110-2-2400 (Headquarters, Department of the Army 1961a). The program considers structures with either one or two bays as shown in Figures 6 and 7. These figures show the geometric outlines and

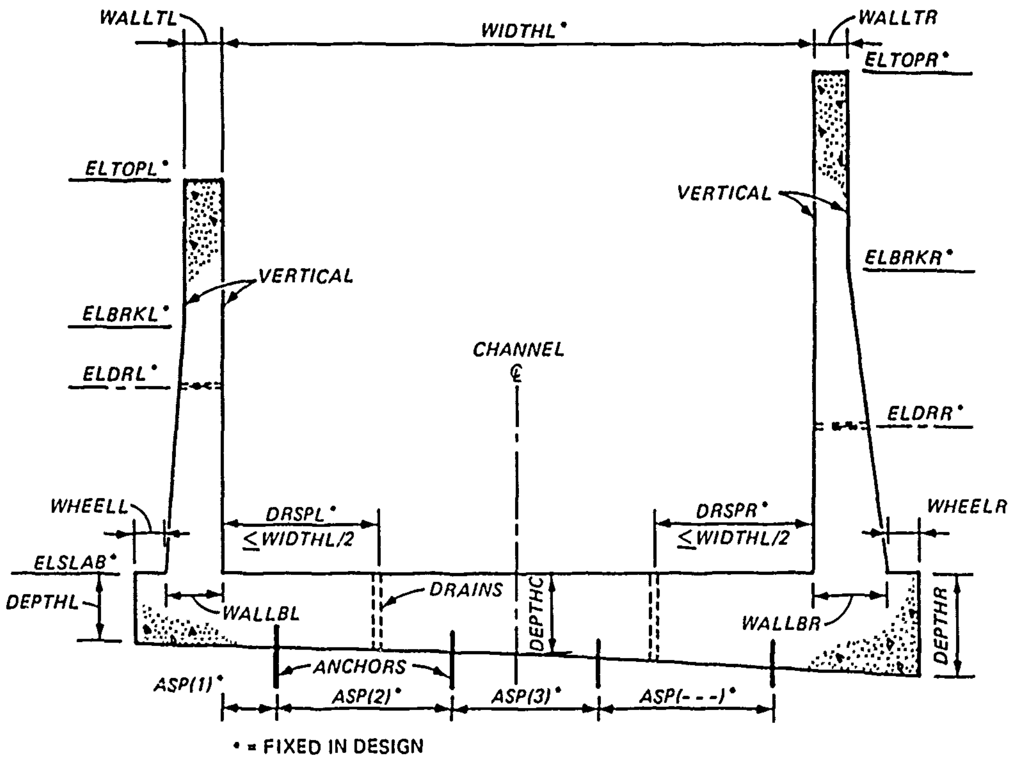


Figure 6. Single-bay channel structure

* A table of factors for converting non-SI units of measurement to SI (metric) units is presented on page 4.

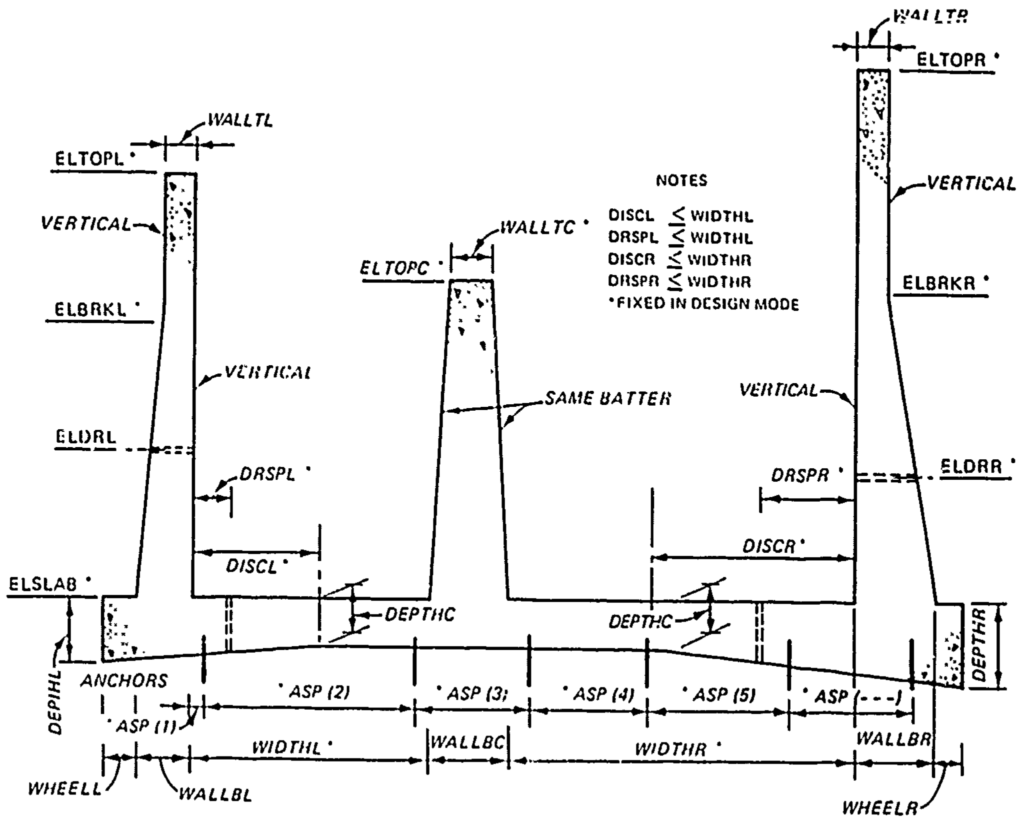


Figure 7. Double-bay channel structure

define the input variables further described in the "User's Guide for Channels" (Volume C of this report). The sections may be unsymmetrical as shown in the figures for the investigation mode. However, the section must be geometrically symmetrical for the design mode. Also, there is a symmetrical input option for the investigation mode which is described in Volume C.

25. The input values define the cross section for the investigation mode. However, in the design mode these values define the initial cross section. Input variables shown with an asterisk are kept constant in the design mode. The cross-section variables not shown with an asterisk are incremented as necessary for the final design. The details of the design procedure are given later in this report.

26. The variables describing the locations of drains and anchors are shown in the figures defining the geometric outlines of the channel. The use of these variables is discussed in subsequent sections; however, several points should be noted about the locations of the anchors by the user. First, the leftmost anchor is located by giving the distance from the inside face of the left wall. This distance may be negative, as long as the anchor remains

under the slab. Subsequent anchors are located by giving a positive distance from the previous anchor.

27. For symmetrical channels, including all channels being designed by the program, only data for the left half of the channel are input. Thus, only the anchors on the left side of the channel are specified, and equal symmetrical anchors are created by the program on the right-half. However, any anchor located within one-tenth of a foot of the center line is treated as a "center-line anchor" and does not cause the creation of another anchor. Also, it should be noted that for the design mode the widths of the walls may increase. Thus, the relative positions of the anchors may shift slightly during the design process. If the walls increase significantly during the design of the walls, the user of the program may respecify the locations of the anchors and rerun the program. Note, however, that the anchor located at the center line using the original minimum wall thickness will remain at that center-line location in spite of the changes in the wall thicknesses.

28. Input and output for the channels are keyed to the members as defined in Figure 8. The details of the frame model are discussed subsequently. However, it is important to note here that the frame analysis considers a frame of relatively flexible vertical and horizontal members connected at essentially rigid joints of finite size. The rigid joints are shown within

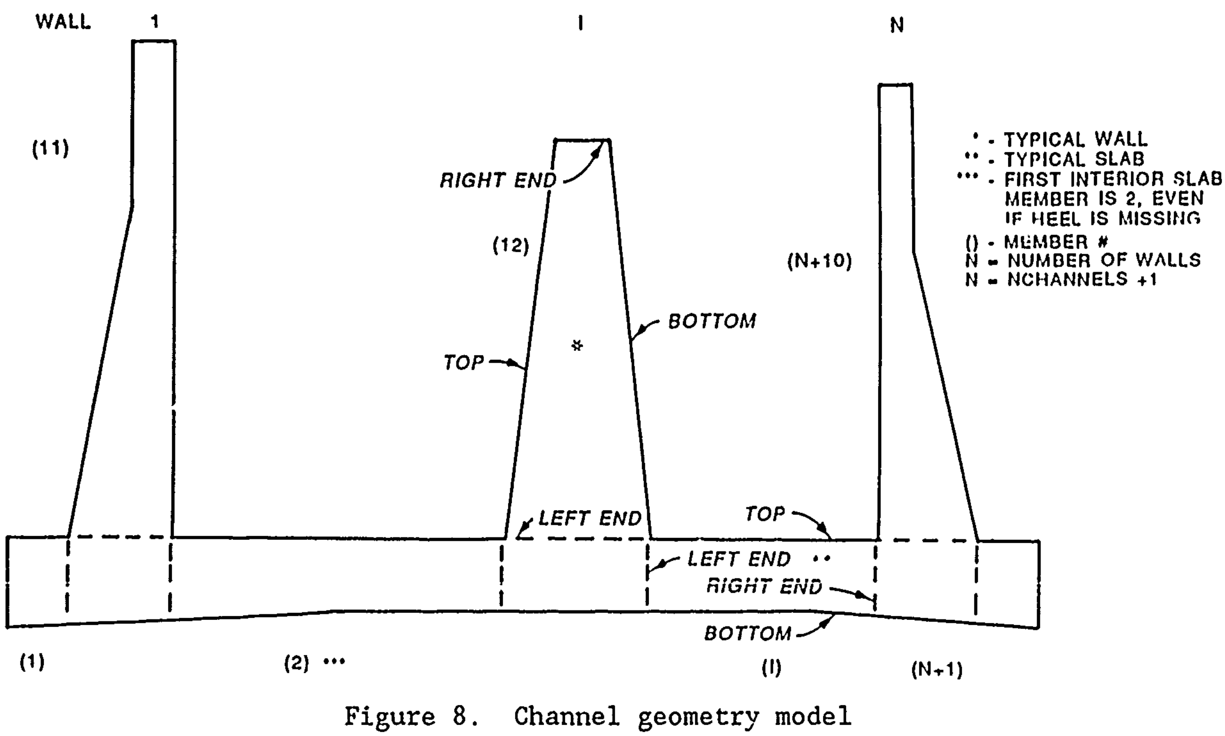


Figure 8. Channel geometry model

dashed lines in the figure. Base slab members including heels are numbered from left to right from 1 to N + 1, where N is the number of walls. The heels may be omitted; however, if the left heel is omitted, the first slab member will still be member number 2. The number of channels is NCHANNELS and

$$N = NCHANNELS + 1$$

The leftmost wall is numbered 11, and then the remaining walls increase in number from left to right as shown in the figure. Input of reinforcing and special loads and the all member output are keyed to these member numbers and the "left-right" - "top-bottom" orientation of the members as shown in the figure. Distances along the member are always specified from the "left" end of the member.

29. As for the basin, section behavior may be reviewed at up to five points per member in the investigation mode. The locations of the review points are specified as shown previously for basins in Figure 5. The distance to the review point is always measured from the "left" end of the member as defined in Figure 5. Up to three layers of reinforcing may be specified for the "top" and "bottom" of a member. As many of the members as desired may be reviewed, NMINV is the total number of members being reviewed. Thus, if all members of a single channel structure with heels were reviewed, NMINV would be five (two walls, two heels, and the center slab).

30. It should be noted by the user that the program will compute stresses or evaluate strength design criteria in the heel in the investigation mode. However, the depth-span ratio of the heels in channels may often exceed that for which the computations are valid. Thus, the user of the program should ensure that the depth-span ratio is sufficiently small for the calculations to be valid.

31. As for the basins, the "top" layers of steel are not considered effective in resisting tension on the "bottom" side of the member. Thus, the user should ensure that steel is located in the proper face for all load conditions.

32. NTOPL and NBOTL are the number of layers in the "top" and "bottom" of the section, respectively. Layers are numbered from the exterior of section to the interior as shown in the figure. The steel within the layers may be specified by two different bar options. For 'REOPT' = "BAR," the steel

within each layer is specified by the bar size (number of nominal one-eighth-in. increments in diameter) and the spacing in inches within the layer. For 'REOPT' = "ARE," the steel is specified by giving the area in square inches per foot of the steel in each layer and the nominal diameter of the steel in the outer layer. This nominal diameter is only used in computing the location of the centroid of the outer steel layer.

33. The variable COVER is the clear cover from the outer edge to the first steel layer and is specified for four different conditions as defined in the "User's Guide for Channels" (Volume C). The center-to-center distance between steel layers, CCLAY, is constant at all locations.

PART III: FRAME ANALYSIS

Frame Analysis Module, FRAME55

34. In order to incorporate limited soil-structure interaction capabilities into the program, it was decided that the frame analysis module should permit frame members to have nonlinear soil support characteristics, i.e. beam on nonlinear elastic foundation. FRAME54 previously developed by the author permits general nonlinear soil supports for members through the use of nonlinear force deformation (q-w) curves describing the lateral and axial forces developed along the length of members as shown in Figure 9. Similar support curves may be specified at the frame joints. Nonlinear stress-strain behavior and nonlinear geometric behavior (buckling and beam-column action) are also modeled in the FRAME54 program.

35. FRAME55 is a modified version of the earlier program eliminating the nonlinear stress-strain and nonlinear geometric models and with other minor modifications to facilitate the specific nature of the U-frame structures. FRAME55 was then made the analysis module of the U-frame analysis program CUFRBC.

36. CUFRBC consists of this frame analysis module, a preprocessor to prepare the voluminous data required by FRAME55 describing the U-frame geometry and loading, and a postprocessor to present the results in a convenient manner, including graphical output.

37. The frame analysis module will only be described briefly herein, since it has been previously documented. As shown in Figure 10, each joint of the 2-D frame has three degrees-of-freedom, and each member has six degrees-of-freedom. Figure 11 shows that the frame members are further subdivided into a number of discrete elements. The discrete element model is an extension of pioneering work on finite difference models by Matlock and Haliburton (1966) and Taylor, Thomas, and Matlock (1968). Details of the discrete element modeling procedure have been well documented (Hays and Matlock 1973; Hays and Santhanam 1979). The use of the discrete elements facilitates the inclusion of variable cross-section properties and all forms of nonlineararity.

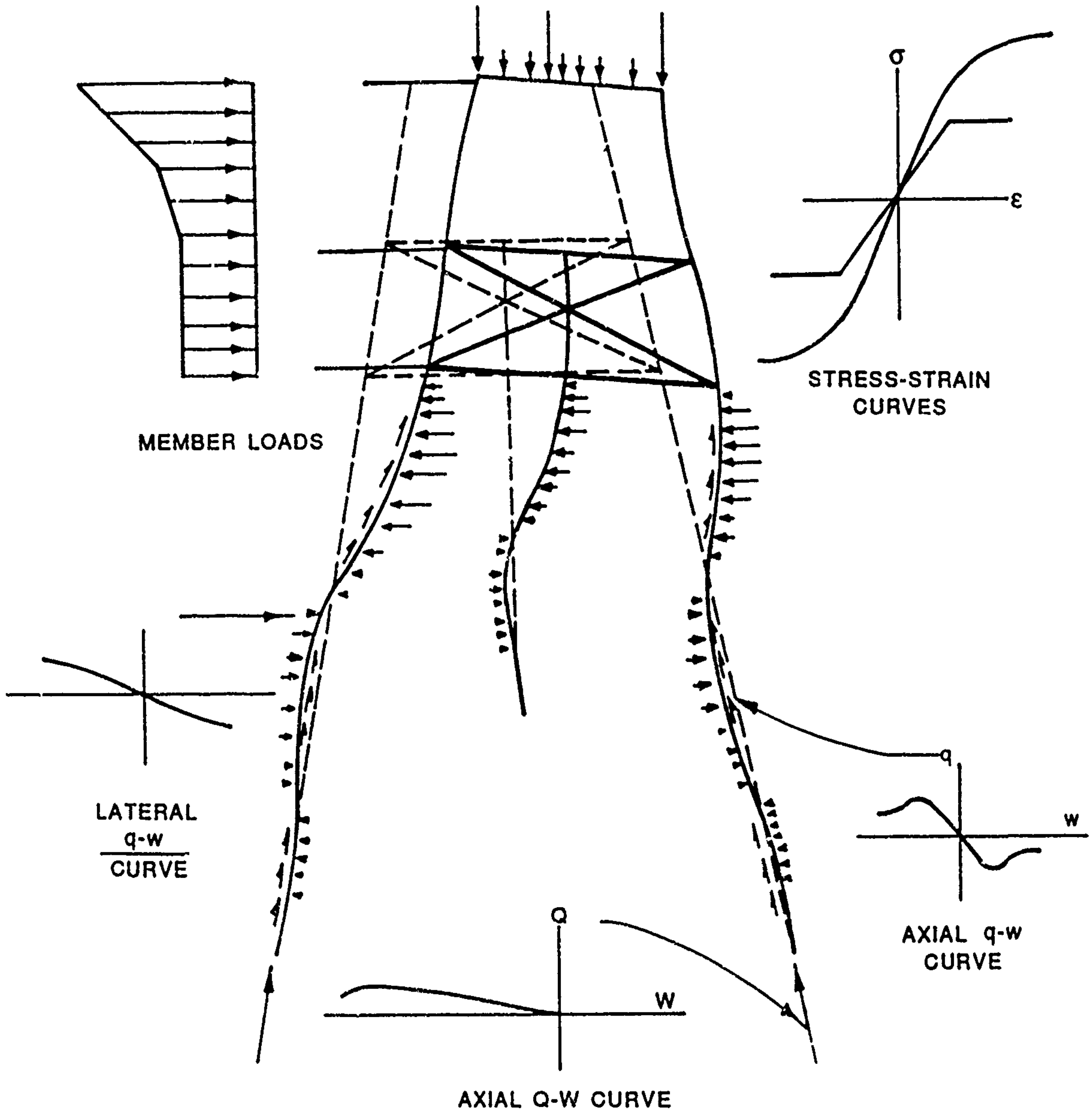
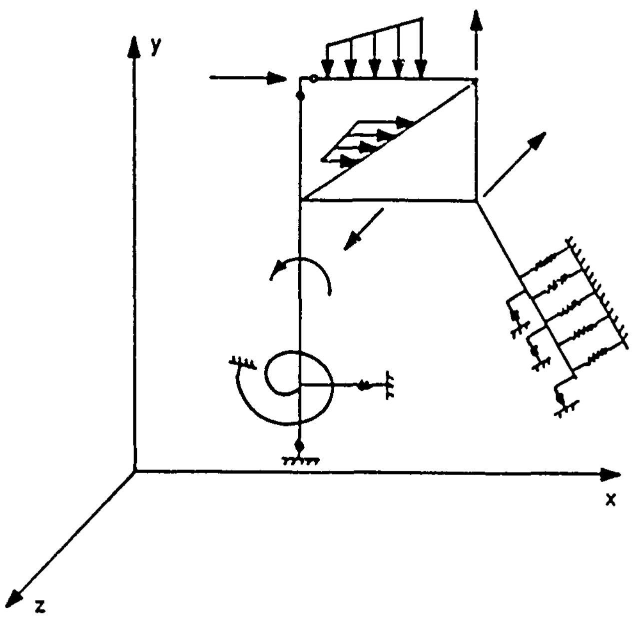
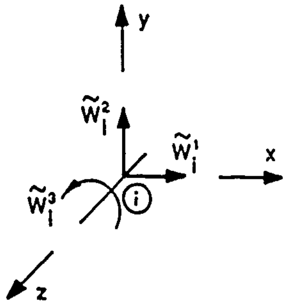


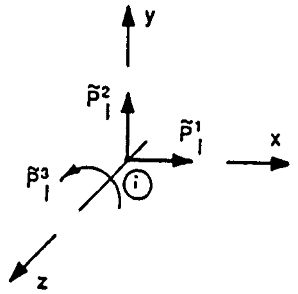
Figure 9. General nonlinear frame behavior



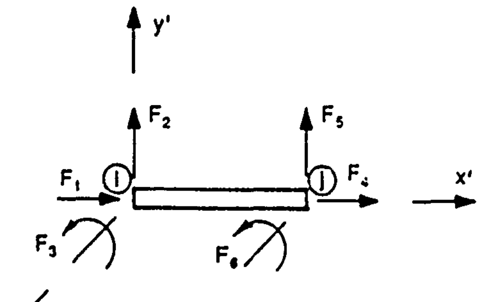
a. Plane frame



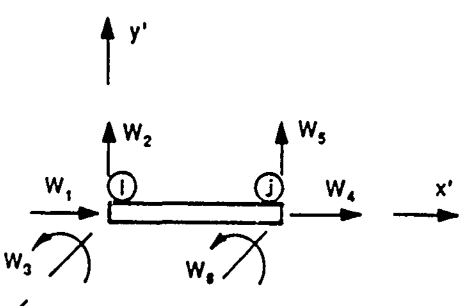
b. Joint displacements



c. Joint forces

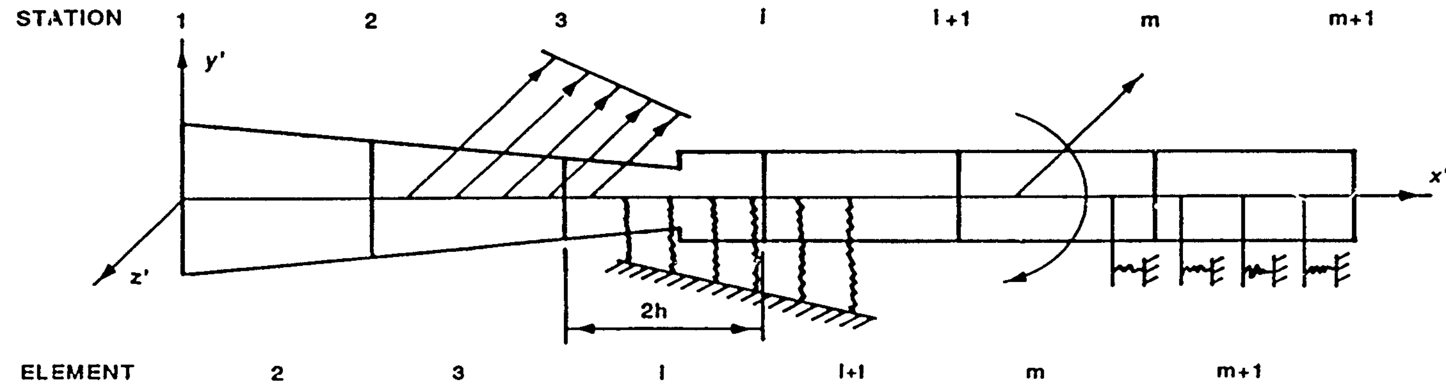


d. Member-end-forces

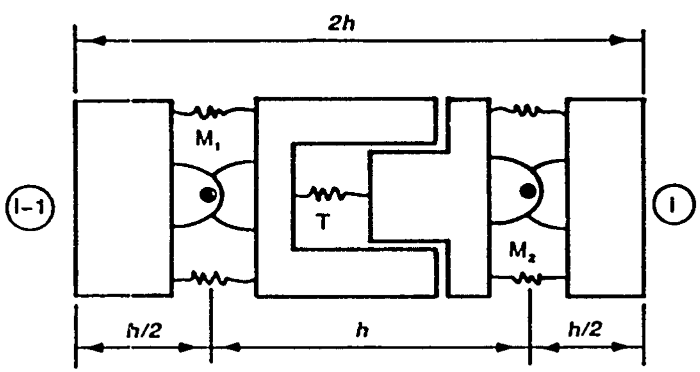


e. Member-end-displacement

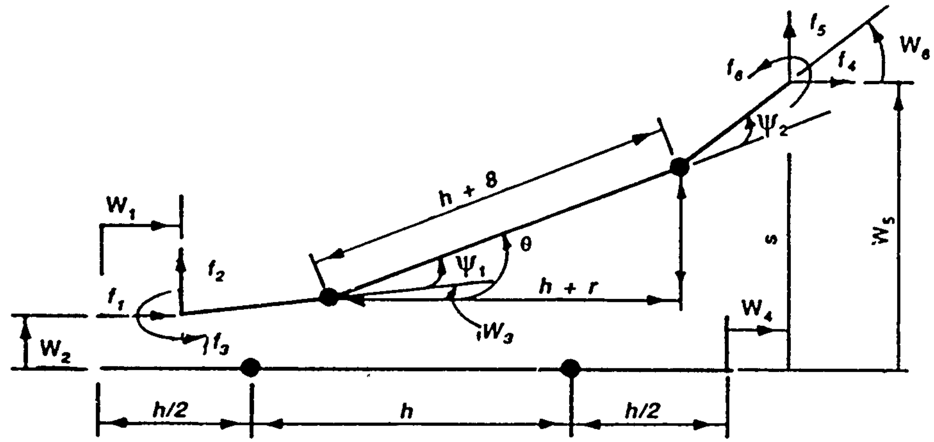
Figure 10. Typical two-dimensional frame model



a. General frame member



b. Mechanical model of element



c. Discrete line element model in undeformed and deformed positions

Figure 11. Discrete element representation of frame member

Frame Model

38. Both the basin and channel structures previously described have many common features, and either can be modeled as a general multiple wall U-frame as shown in Figure 12. Thus, it was decided to write one program that would handle both structure types. However, separate user's guides (Volumes B and C) are available for the two structure types. Further, once the user of the program specifies which type of structure is being analyzed or investigated, the program blocks out all references to input other than for the user's type of structure. Likewise, if the person using the program is working with a single-bay structure, input references and output for other portions of larger structures will be omitted.

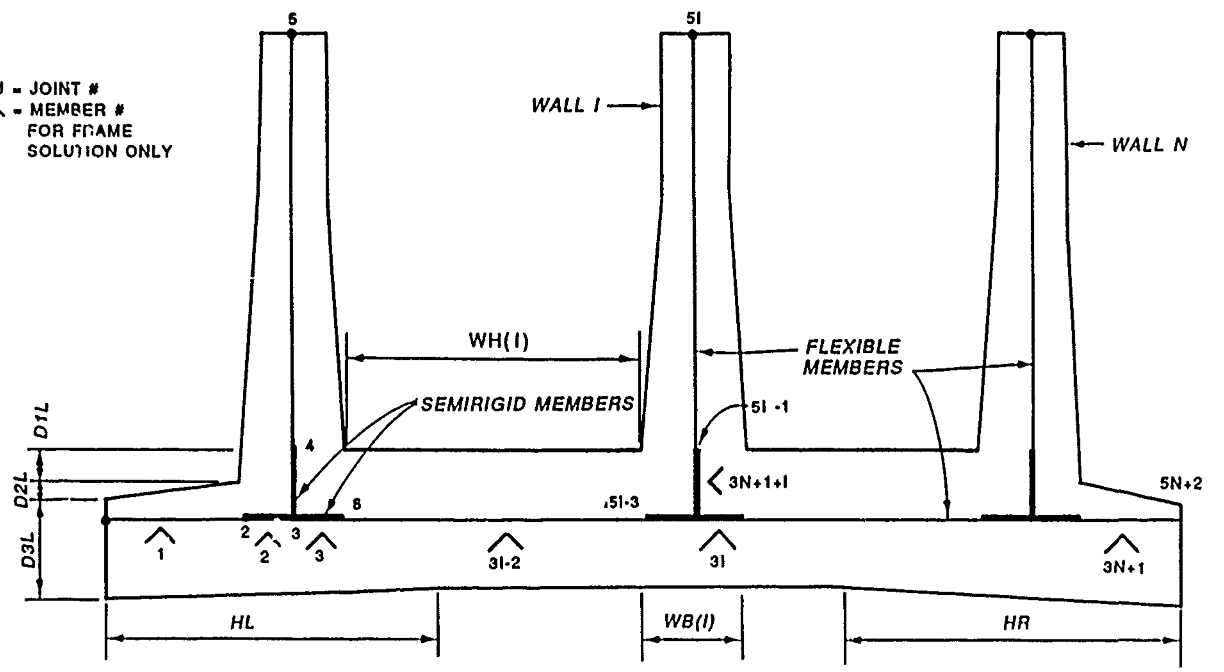


Figure 12. U-frame geometry model

39. The general U-frame model shown in Figure 12 is seen to encompass both the basin and channel frame models previously described. The maximum number of walls permitted is four for the basin and three for the channel. The geometry variables shown in Figure 12 are computed in terms of the input variables previously shown for the two structure types. Thus, the more detailed computations in the program were programmed only once and will cover both structure types.

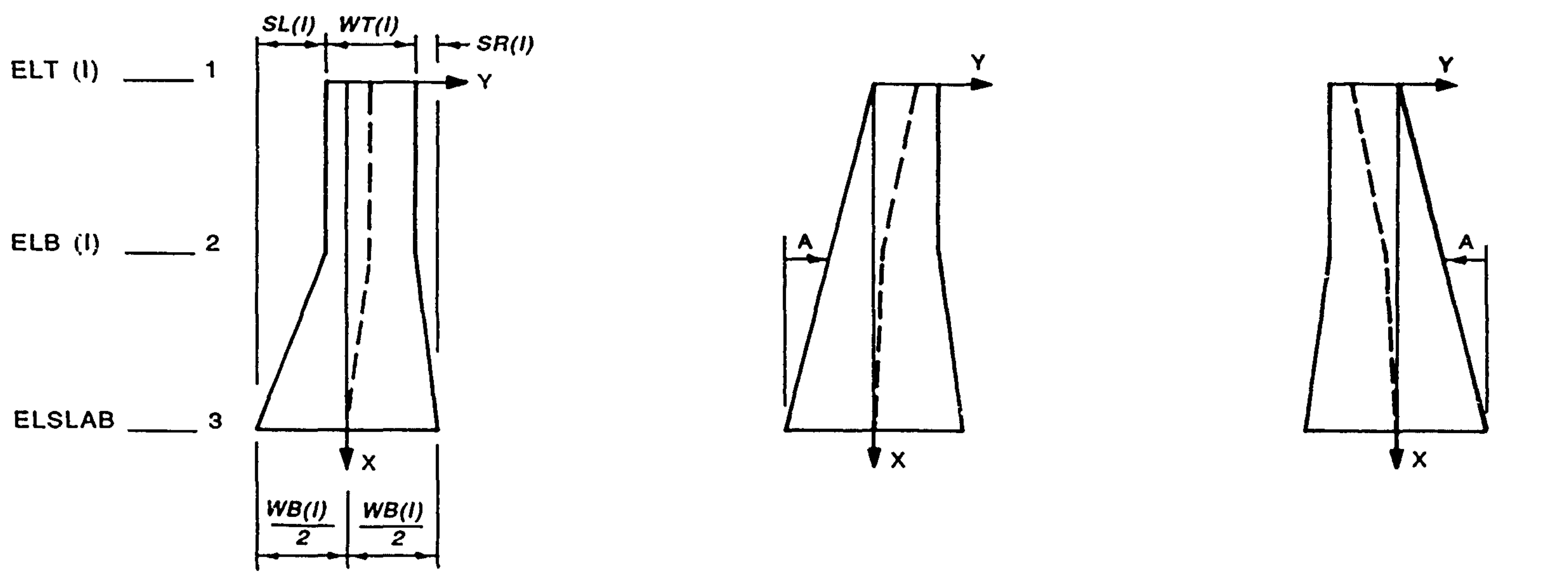
40. The frame members are taken as essentially vertical and horizontal. The idealized axis for all the horizontal members is taken at the middepth of the central portion of the slab. Similarly, the idealized axis of all wall members are taken at the center of the walls at the elevation of the top of the slab. The eccentricity of the centroid of the cross section from the idealized axis is however considered.

41. Figure 12 shows the member numbers used in the frame solution. It will be seen that these numbers differ from those previously assigned for program input and output. However, the member numbers used for the frame solution are not needed for the use of the program and are only shown here to illustrate the frame model used in the analysis. The reason that the numbers differ is that the essentially rigid joint regions of the structure are modeled by defining members that have an area equal to twice the actual depth and a moment of inertia equal to eight times the actual value. Thus, the deformations in these semirigid members are insignificant compared to those of the more flexible portions of the frame. Frame geometry data for the frame analysis module (joint coordinates and member incidences) are automatically generated by the program from the geometric variables shown in Figure 12.

Member Geometry and Stiffness

42. Figure 13 shows typical wall members. All types of basin and channel walls previously described may be handled by these three wall types. The program computes the general wall variables shown in Figure 13 in terms of the geometric input for the basin or channel structure as appropriate. Next, the wall thickness and distance from the idealized wall axis to the centroid of the wall at the critical points are computed as shown by the equations on the bottom of the figure. The orientation of the member or local centroidal X-axis is from the top to bottom of the walls. This orientation is only used internally in the FRAME55 module and should not be confused with the input and output distances which are all specified from the "left" end or bottom of the walls.

43. Calculations for the depth and centroidal offsets are also made, in a similar manner, for the base slab members using the general U-frame geometric variables shown in Figure 12. Internally, the flexible wall and slab members are subdivided into ten discrete elements as shown in Figure 11. The



KW(I) = 1

DI = WT(I)
YI = [WT(I) - WB(I)]/2+SL(I)
D2 = DI
Y2 = YI
D3 = WB(I)
Y3 = O

D = DEPTH OR THICKNESS
Y = DISTANCE FROM REFERENCE AXIS TO CENTROID

KW(I) = 2

D1 = WT(I)
YI = [WT(I) - WB(I)]/2 + SL (I)
D2 = WB(I) - A - SR (I)

$$A = SL(I)\,\frac{[ELB(I) - ELSLAB]}{[ELT(I) - ELSLAB]}$$

Y2 = A + [D2 - WB(I)]/2
D3 = WB(I)
Y3 = O

KW(I) = 3

DI = WT(I)
YI = "DITTO"
D2 = WB(I) -A - SL(I)

$$A = SR(I)\,\frac{[ELB(I) - ELSLAB]}{[ELT(I) - ELSLAB]}$$

Y2 = S L(I) + [D2 - WB(I)]/2
D3 = WB(I)
Y3 = O

Figure 13. Wall member geometry

semirigid joint members are each subdivided into four elements. The depth and centroidal offsets at the longitudinal midpoint of each element are computed by interpolation of the depths and centroidal offsets at the critical points in the members.

44. The gross cross-sectional properties A,I, and AY are next computed at the midpoint of all elements where A is the area of a 1-ft-wide strip of the U-frame member, I is the corresponding moment of inertia about the member idealized axis, and AY is the product of the area times the distance the centroid of the section is offset from the idealized member axis. The modulus of elasticity, EC, in units of kips per square inch is taken as constant using the American Concrete Institute (1983) equation:

$$EC = 33.*WCEFF^{1.5}*FPC^{0.5}$$

where FPC is the compressive strength in pounds per square inch, and WCEFF is the effective unit weight of the concrete in pounds per cubic foot. WCEFF is computed by subtracting 6 pcf from the input unit weight of the concrete to account for the weight of the steel reinforcement.

45. Gross section properties are used throughout the analysis, since generally stresses are kept low enough in basin and channel structures to avoid significant cracking. If the stresses should be high enough to cause cracking, the deflections computed by the program would be too low. Likewise, no allowance for creep is made in the analysis for deflections.

PART IV: PROGRAM LOADING OPTIONS

Nature of Loading

46. The U-frame structure is basically a long, open top cellular concrete tube inserted into a soil-water medium. The primary loadings on the U-frame come from the geohydraulic pressures generated against the surfaces of the tube as they restrain the soil and water. These pressures are very dependent on the construction techniques used in excavating and backfilling, the geometry and stiffness of the U-frame, and the elastic and plastic properties of the soil.

47. The U-frame structure must function in a variety of flow conditions from drought to flood. Thus, the hydraulic loading varies considerably and may be transient in character. Standard design procedures have developed based on analyzing a 1-ft slice of the U-frame. This planar model of the U-frame is analyzed for a variety of approximate loadings. The exact nature of the loading or the physical parameters on which the loadings are based are never known precisely. Thus, the designer is forced to look at extreme ranges of possibilities and determine a range of loadings which control the size of the U-frame cross section and the reinforcing at various points within the section.

48. The nature of the loading is such that all the geohydraulic forces are based on an interaction between the U-frame section and the soil-water medium. To rigorously analyze the structure for the true interaction is perhaps technically feasible today through 3-D finite element models. However, it remains to be seen if such models will produce designs that are more economical or statistically safer than traditional methods.

Active and Reactive Loading

49. For the planar models of analysis, it is convenient to subdivide the loadings on the structure into two primary classifications, active and reactive loads. Active loadings are primarily those that tend to move the U-frame structure, and reactive forces are those that are developed to counteract or oppose that motion. Hydraulic forces will always be considered as active loadings as will be any special loads that may be specified by the user

such as wind forces. The self-weight of the structure is another active loading.

50. Soil pressures or forces may be either active or reactive. Examples of active pressures are the active earth pressure on the U-frame walls and the weight of the soil on top of the heels of U-frames. The base foundation pressure developed as the foundation moves into the soil is an example of reactive pressure.

51. For unsymmetrical U-frames, the pressures and forces developed to maintain horizontal equilibrium with the active pressures and forces are reactive and will include base shears and lateral earth pressures which may approach the passive state. The soil forces or loading are actually quite dependent on the deformation of the frame and the soil mass, and a limited form of soil-structure interaction may be modeled through the use of nonlinear force-deformation curves. While these curves may be used to model the full range of loading between active and passive conditions, the forces they represent will be classified as reactive forces in discussing this program.

52. The program CUFRBC computes the different types of active forces and pressures to be developed against the surfaces of the U-frame. Then, in general, a frame analysis is made for the frame subjected to these loadings to find the reactive forces and the internal force distributions of shear, axial force, and moment for design. Actually, in some of the options, certain of the reactive loadings may be computed and applied prior to using the frame analysis module.

53. The program provides for a wide variety of different ways of specifying the loadings in order to allow different design practices to be followed using the same program. Thus, the program can be used to make important parameter studies comparing various design approaches. Also, while the program is quite comprehensive, the input is still simple enough such that a designer will be able to use the program efficiently for routine designs that may use only a small portion of the allowed program options. However, it is recommended that anyone planning to use the program read the descriptions of all the possible loadings before attempting to apply the program.

Description of Geohydraulic Loads

54. Figure 14 shows soil, water, and rock elevations and surcharge data

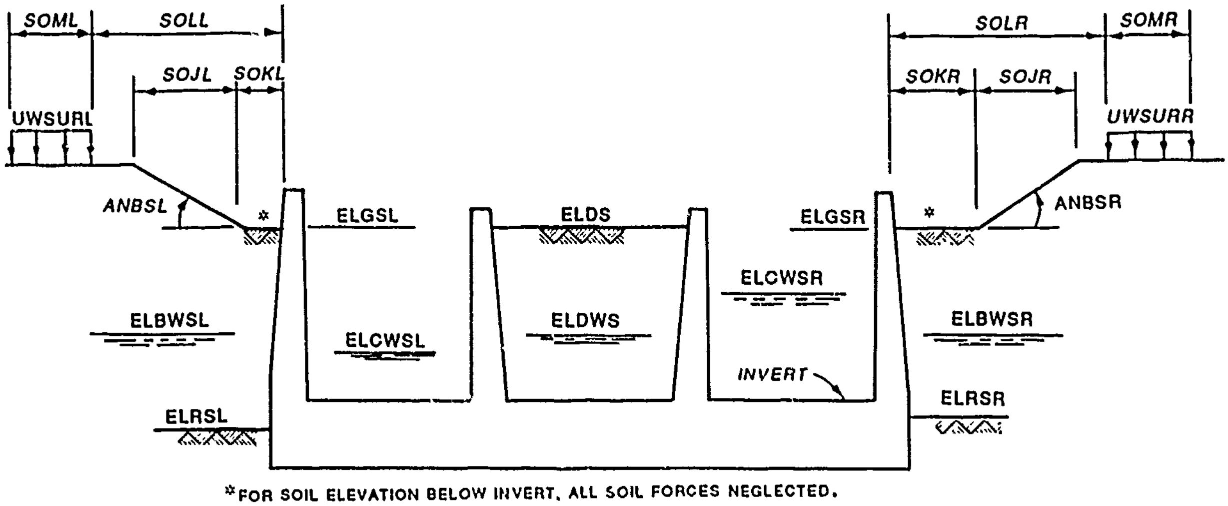


Figure 14. Ground profile, water elevations, and surcharge

which are input for a general U-frame structure. Of course for a one- or two-bay structure, certain of the items are omitted. The user's guides (Volumes B and C) specify which of these items are required for the particular structure geometry.

55. The various types of active and reactive loadings are next briefly reviewed. Then the loadings are described in detail. Some of the loadings described cannot be used simultaneously in the program. For instance, either empirical wall pressures or wedge solutions may be used but not both within the same computer solution. Thus, after all loadings are described, the various program options concerning loading are discussed in the section entitled "Program Loading Combinations." Certain of the loading options are not permitted in the design mode. The design loadings are generally restricted to symmetrical cases. Details on the loadings for the design mode are covered subsequently in detailed discussions of the design mode.

Summary of Active Loadings in Investigation Mode

56. The CUFRBC program allows for the following types of active loading in the investigation mode:

a. Self-weight of concrete U-frame automatically generated from the geometry of the section and the input unit weight for all load conditions.

b. Hydraulic loading wherein all hydraulic pressures are automatically computed from the input water elevations, drain locations, and specified drain efficiencies.

c. Active earth pressure by wedge solution. A wedge solution may be performed to give active earth pressures for symmetrical soil loadings. For unsymmetrical situations, the pressure on the active side may be obtained by an active wedge solution.

d. At-rest pressures by multiplying input coefficient times active earth pressures.

e. Vertical surcharge loads as part of wedge solution.

f. Empirical wall and heel pressures computed from input soil elevations and lateral pressure coefficient.

g. User specified special loads. General concentrated and distributed loads are at any points along the section. These loads may be used to represent types of loadings other than those generated directly by the program. Also, the special loads can be used to "correct" any loading that the program computes in a different manner than that normally done by the user. The special loads may be combined with any of the other active and reactive loads.

Summary of Reactive Loadings in Investigation Mode

57. The CUFRBC program allows for the following types of reactive loading in the investigation mode:

a. Base slab pressures computed using compression only beam on elastic foundation model, i.e., distributed vertical elastic springs acting only in compression.

b. Vertical tiedown forces computed as tension only elastic spring forces.

c. Base slab pressures computed by statics with user specified shape. This method is similar to a "P/A" $\pm$ "Mc/I" approach except the shape of the "P/A" portion can be specified.

d. Base shears computed to satisfy the horizontal equilibrium from having all active forces be either uniformly distributed over the base or on the basis of distributed horizontal springs on the base slab.

e. Lateral wall pressures on both active and passive sides computed using nonlinear force-deformation curves and the compatibility of deformation with wall deflection. These so-called q-w curves may be input to range from the full active to passive states.

f. Base shears and earth pressures on the passive side of U-frame based on the proportional distribution of potential maximum passive values, primarily for nonsymmetric loadings.

Hydraulic Loading

58. The hydraulic loading on the structure is automatically computed with the assumptions described herein. The calculations do not follow the line of the creep theory as outlined in EM 1110-2-2502 (Headquarters, Department of the Army 1961b). However, the pressures will not differ much from the line of creep calculations, and users may adjust the computed pressures or give their own hydraulic pressures by including the special loads option.

59. The hydraulic pressures acting on the U-frame are computed in terms of the effective water elevations, ELW(I), adjacent to each wall as shown in Figure 15. The actual water elevations are input as ELBWSL, ELCWSL, ELDWS, ELCWSR, and ELBWSR, shown in the figure. The actual elevations are input as necessary for the particularly structure (basin or channel) and with consideration of symmetry as described in the user's guides.

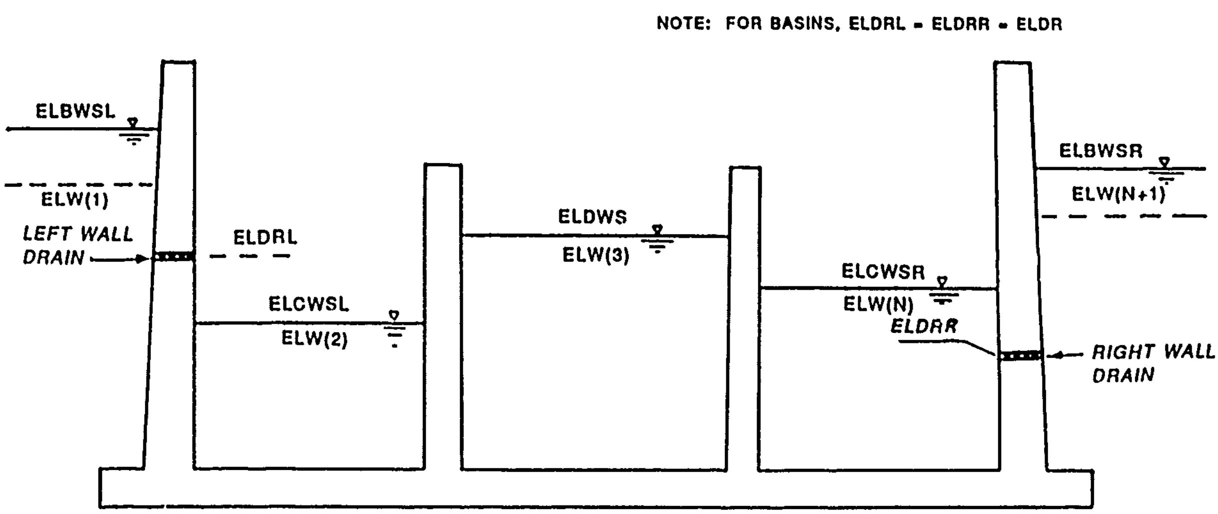


Figure 15. Input and effective water elevations

60. The effective interior water elevations are simply the corresponding input values. However, the effective exterior water elevations, ELW(1) and ELW(N+1), are computed considering the percent effectiveness for the exterior wall drains. The exterior wall drains are only considered effective in draining water into the U-frame and thus only affect the exterior effective water elevations. The interior water elevations are not affected by the wall drains.

61. It should be noted that since the wall drain option in effect only

lowers the exterior wall elevations, the same results as using the wall drain option could be obtained by simply setting the exterior water elevations at their effective values. However, the wall drain option was included to allow automatic reduction of the exterior elevations based on input values of drain effectivenesses. The percent effectiveness operates on the smaller of the difference in head between the exterior water elevation and the wall drain or the exterior and interior water elevations as illustrated below.

62. First, consider the case where the wall drain is above the corresponding interior water elevation as illustrated in Figure 15 for the left wall drain. If the percent effectiveness for the left exterior drain is PDRNWL and if ELBWSL is greater than ELDRL, the effective elevation, ELW(1), is computed by

$$ELW(1) = ELBWSL - PDRNWL*(ELBWSL - ELDRL)/100$$

unless ELBWSL is less than ELDRL, in which case the drain is not considered and ELW(1) is equal to ELBWSL.

63. Next, consider the case where the wall drain is below the corresponding interior water elevation as illustrated in Figure 15 for the right wall drain. If the percent effectiveness for the right exterior drain is PDRNWR and if ELBWSR is greater than ELCWSR, the effective elevation, ELW(N+1), is computed by

$$ELW(N+1) = ELBWSR - PDRNWR*(ELBWSR - ELCWSR)/100$$

unless ELBWSR is less than ELCWSR, in which case the drain is not considered and ELW(N+1) is equal to ELBWSR.

64. Of course, both of the above cases are checked for both the left and right exterior walls. Also, the user may omit all input for the wall drains if desired, and the effective water elevations for the exterior walls will be those input. It should be noted that the effective elevations for the exterior wall are used not only in computing wall pressures but also in computing uplift pressures on the base in conjunction with slab drains.

65. Hydraulic forces on the wall members are computed at the center of each of the 10 discrete elements of length TH in the frame model as shown in Figure 16. The resultant force PR acts normal to the wall, and the vertical

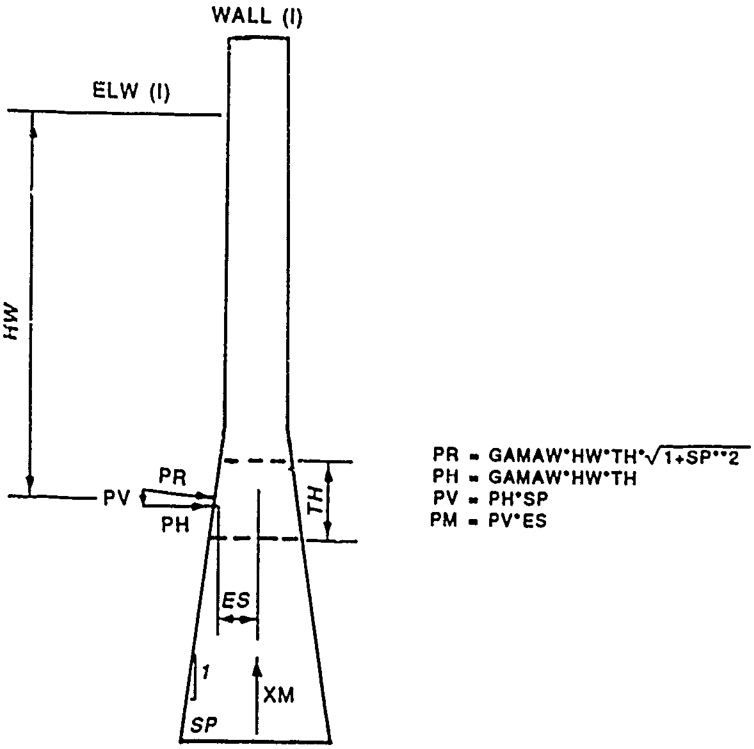


Figure 16. Hydraulic forces on one side of wall

and horizontal forces and the moment of the vertical force are computed as shown in the figure. Similar computations are made for both sides of the wall, and the forces summed to obtain the net hydraulic forces.

66. The hydraulic forces acting on the base slab are computed in a similar manner. However, first, the effective head along the bottom of the slab must be found with due consideration of the drains. The procedure for computing the effective head at each of the drains is illustrated in Figure 17.

67. First, the reference head, EHB, is computed at each of the drains. EHB is the head that would be acting assuming no drain effectiveness and a linear variation of head across the base. The head on the top of the slab, EHT, and the head from the water on top of slab projected to the bottom of the slab, EHTP, are next found from the water elevations ELW(I). Then the effective head at drain J, EH(J), is found by

EH(J) = EHB - PDRAIN(J)/100*(EHB - EHTP)

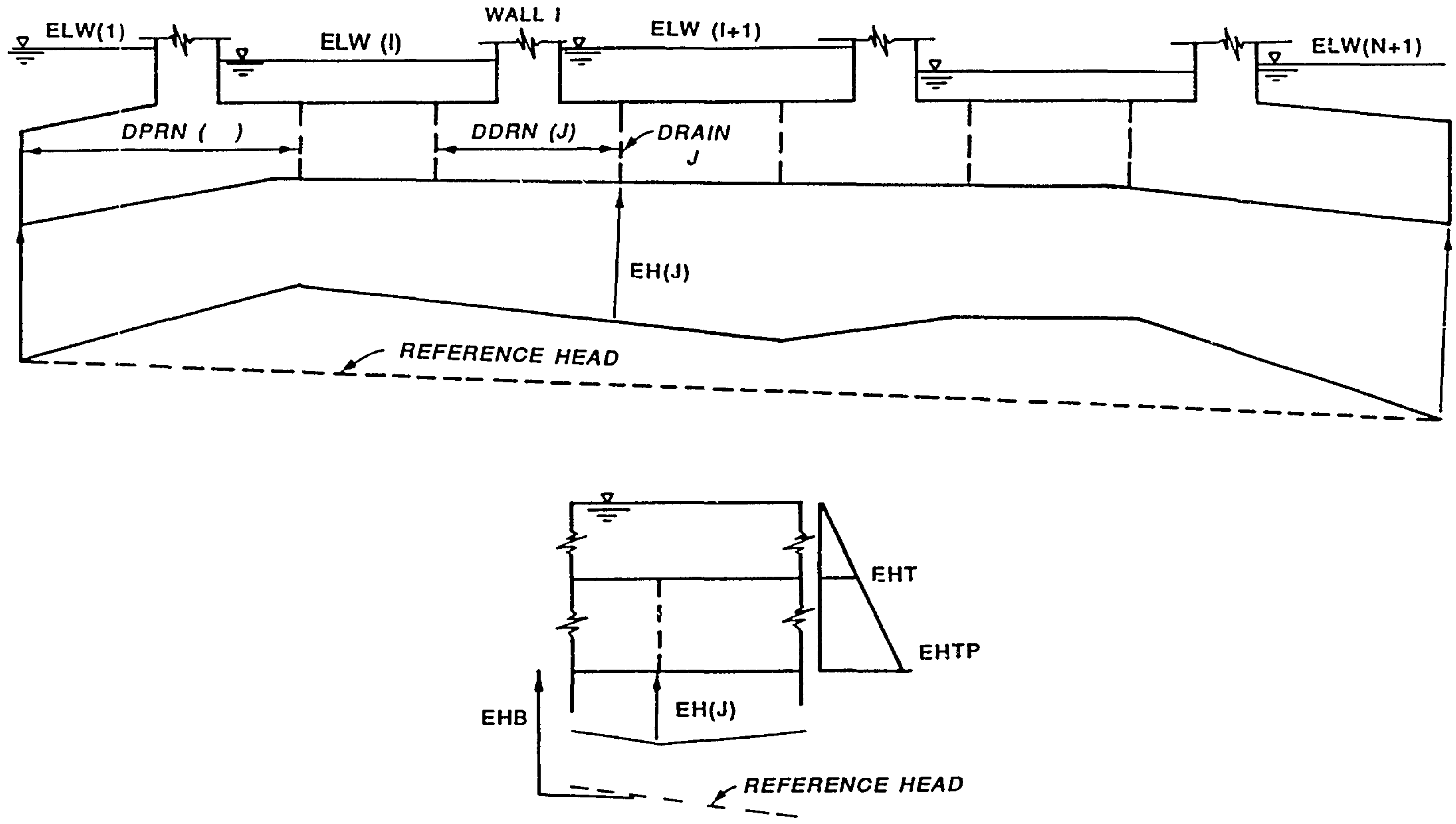


Figure 17. Hydraulic forces on base slab

where PDRAIN(J) is the percent effectiveness of the vertical drain. The head on top of the slab is not adjusted for the effectiveness of the slab drains; however, if EHTP is greater than EHB, and the drain is considered, the water pressure on the base will be increased. For the channel structure, the head is corrected to account for the fact that the base is not a straight line all across the bottom.

68. If a drain is specified as 100 percent effective, then the head on the bottom of the slab at the drain will be EHTP with the head on top of the slab based on EHT. If the drains are specified as being 0 percent effective, then they have no effect on the hydraulic forces. Further, a drain option is specified which allows the user to avoid all input of slab drain data.

Active Pressures Using Wedge Solution

69. Active pressure is based on a condition of limit equilibrium. The soil forces acting on the faces of the walls and the top of the base slab may be obtained from the active wedge solution described herein. The solution differs slightly from that used in standard stability analysis, because it was formulated to give the distribution of forces acting on the faces of the U-frame.

70. Figure 18 illustrates how the wedges are solved incrementally to give the required force distribution. Up to 10 different wedges are taken along the face of the wall with the bottom of each wedge corresponding to the tenth points, vertically from the top to the bottom of the wall. The figure also shows a free body of a typical Ith wedge. It is desired to find the incremental force PUN(I) which is assumed acting at the midpoint of the segment. It is assumed that similar solutions have already been made for the previous I-1 wedges. Thus, the forces PXN and PYN are already known from summing the vertical and horizontal components of the previous incremental forces up to PUN(I-1).

71. The force PUN(I) makes an angle α with the horizontal. If wall friction is ignored, then α is the angle the wall makes with the vertical, αw. If it is desired to account for a wall friction angle δf, then $\alpha = \alpha w + \delta f$. Trial wedge solutions are made for various values of the angle Θ, the unknown ray of the soil that the wedge makes with the vertical. W is the effective

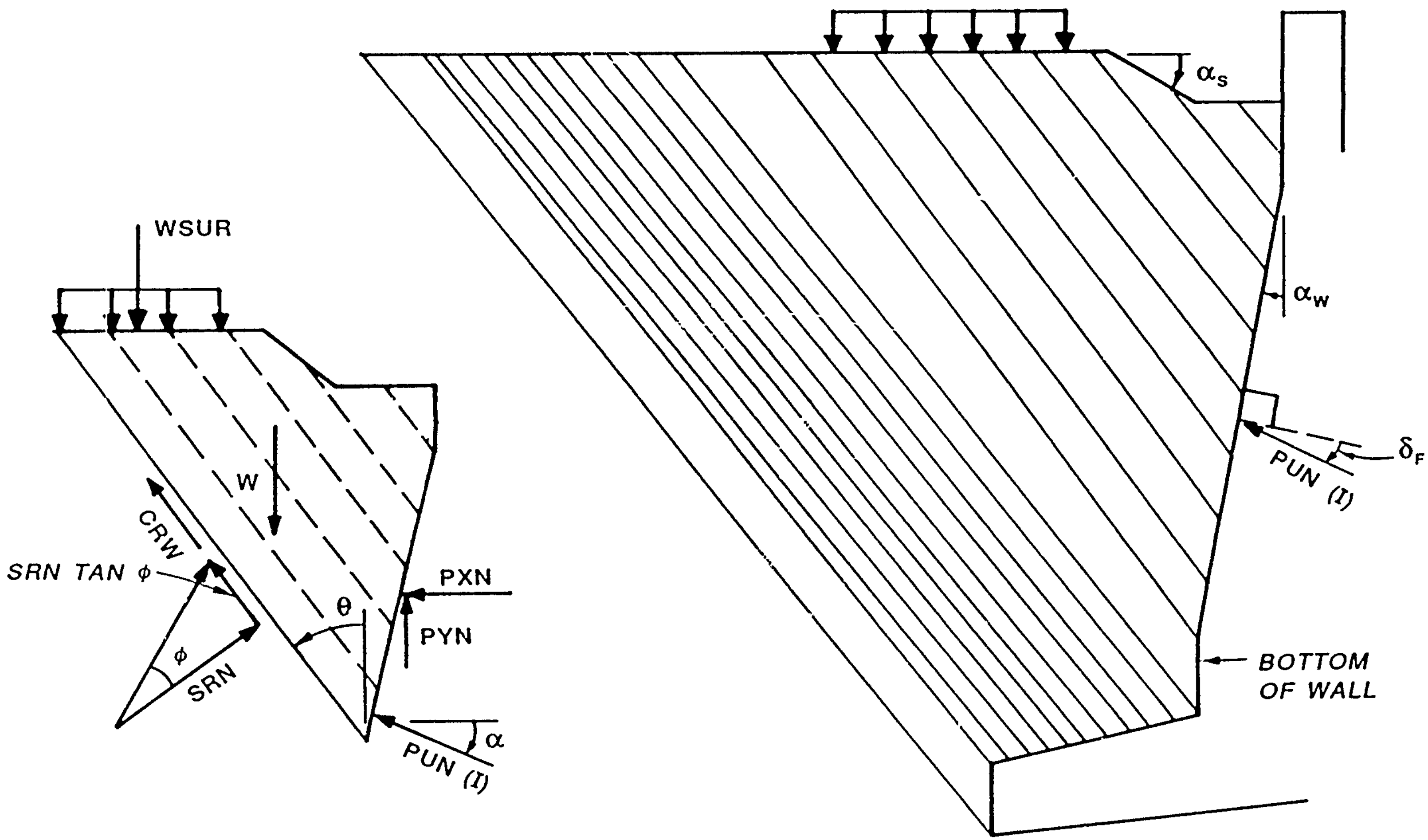


Figure 18. Active wedge solution

weight of the soil in the trial wedge. WSUR is the weight of the surcharge contained in the trial wedge.

72. The other forces acting on the trial wedge are a normal force SRN and two shear forces, CRW and SRN*tan(ϕ), where ϕ is the angle of internal friction for a cohesionless soil. The shear force CRW is due to cohesion, if any, and is found by multiplying the cohesive stress times the length of the ray. The shear forces along the unknown ray are the forces that are a function of the soil properties and can be multiplied by the strength reduction factor SRF to take the variation of soil properties into account. This multiplication is done in the equations that follow for generality. However, the value of SRF is set equal to one in the program.

73. From the summation of vertical forces equal zero

$$\mathrm{SRN}*\sin(\theta) + \mathrm{SRF}*[\mathrm{SRN}*\tan(\phi) + \mathrm{CRW}]*\cos(\theta) + \mathrm{PYN} + \mathrm{PUN}*\sin(\alpha) = \mathrm{W} + \mathrm{WSUR}$$

Thus,

$$\mathrm{SRN} = [\mathrm{W} + \mathrm{WSUR} - \mathrm{SRF}*\mathrm{CRW}*\cos(\theta) - \mathrm{PYN} - \mathrm{PUN}*\sin(\alpha)]/\mathrm{DEN1}$$

where

$$\mathrm{DEN1} = [\sin(\theta) + \mathrm{SRF}*\tan(\phi)\cos(\theta)]$$

From the summation of horizontal forces equal zero

$$\mathrm{SRN}*\cos(\theta) - \mathrm{SRF}*[\mathrm{SRN}*\tan(\phi) + \mathrm{CRW}]*\sin(\theta) = \mathrm{PXN} + \mathrm{PUN}*\cos(\alpha)$$

Thus,

$$\mathrm{SRN} = [\mathrm{SRF}*\mathrm{CRW}*\sin(\theta) + \mathrm{PXN} + \mathrm{PUN}*\cos(\alpha)]/\mathrm{DEN2}$$

where

$$\mathrm{DEN2} = [\cos(\theta) - \mathrm{SRF}*\tan(\phi)*\sin(\theta)]$$

Equating the two expressions for SRN yields

$$\mathrm{PUN} = \{[\mathrm{W} + \mathrm{WSUR} - \mathrm{SRF}*\mathrm{CRW}*\cos(\theta) - \mathrm{PYN}]*\mathrm{DEN2} - [\mathrm{SRF}*\mathrm{CRW}*\sin(\theta) + \mathrm{PXN}]*\mathrm{DEN1}\}/[\sin(\alpha)*\mathrm{DEN2} + \cos(\alpha)*\mathrm{DEN1}]$$

74. The wedge is solved for various trial values of θ to obtain the maximum value of PUN(I) using the half-interval method. Figure 19 shows a flowchart for the half-interval method. The flowchart is set up in general mathematical terms for finding the maximum of a function f(x). Actually f(x) is PUN(I), and x represents the unknown angle θ. XL and XR are the lower and

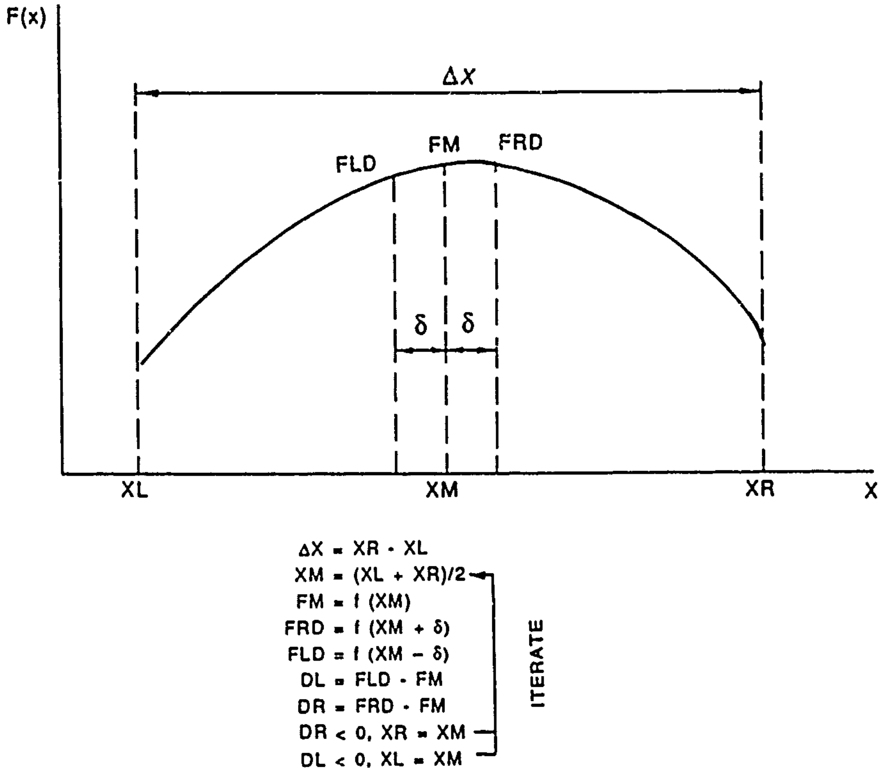


Figure 19. Half-interval method for maxima

upper limits on the search for the unknown value of x or θ. In the program, the lower and upper limits on the search for θ, in degrees, are set at 0.5 and the smaller of 89.5 - ϕ or 89.5 - αs, where αs is the angle the ground makes with the horizontal as shown in Figure 18. Experience showed that maxima would not occur for angles less than 0.5 or greater than 89.5 - ϕ. The limit of 89.5 - αs was set to avoid spurious solutions for cohesive soils.

75. After the force PUN(I) is found, it is broken up into horizontal and vertical components and combined with the hydraulic forces before the frame solution is made. Next, a similar trial wedge solution is found to solve for the forces on the vertical face of the wall below the invert elevation. Then 10 trial wedge solutions are made for forces on the top face of the heel, and finally a trial wedge solution is made to find the force on the vertical face of the heel. All the trial wedges solutions follow the same procedure as described for the wall with PXN and PYN being set equal to the sum of the x and y components of the forces found from the prior wedges. However, if a wall friction angle is specified, it is not used for the wedges solved for the heel.

76. Figure 20 illustrates the geometric procedures used in calculating the weight of the soil mass in the trial wedges. Two cases are considered involving the soil elevation relative to the wall. The soil may intersect the wall either above or below the break in the slope of the wall. However, if the elevation is below the invert elevation, the weight of the soil is ignored and no wedge solutions are made. The volume of the soil is computed using the traverse formula for surveying calculations in terms of the coordinates of the traverse of points outlining the soil wedge. The intersection of the groundline and the wall is numbered as point 0 and given the coordinates of 0,0 for convenience. Point NPW is the number of the point at which the unknown soil ray starts. Point NPW will of course be at a different location for each of the incremental wedges that are solved.

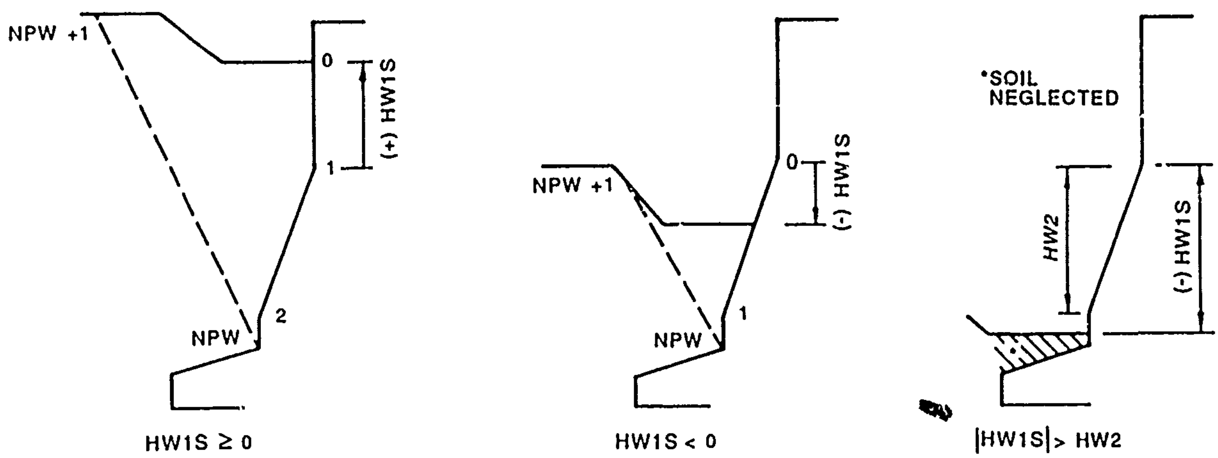


Figure 20. Transverse points on wall

77. Figure 21 shows how the intersection of the soil ray and the ground line is computed for the three cases shown. The X and Y values computed are for point NPW + 1. After this point is located, the remaining points in the closed traverse can easily be found.

78. Figure 22 illustrates the combinations considered in computing the effective weight of the soil for various combinations of water elevations and soil geometries. Note that if the water elevation is below the invert, then the drained unit weight is used for all the soil in all trial wedges.

79. The volume of the saturated soil is computed by a traverse solution similar to the one used for the total soil volume. Then the volume of the drained soil is computed by taking the difference between the total volume and

1 - $Y = 0$, $X \leq SW1$

2 - $Y = SW3/SW2\,(X - SW1)$, $X \leq SW1 + SW2$

3 - $Y = SW3$

$$Y = Y(NPW) + [X - X(NPW)]/\tan\theta$$

1 - $X = X(NPW) - Y(NPW)\tan\theta$

2 - $$X = \frac{\dfrac{Y(NPW) - X(NPW)}{\tan\theta} + \dfrac{SW3 * SW1}{SW2}}{\dfrac{SW3}{SW2} \quad \dfrac{1}{\tan\theta}}$$

3 - $X = [SW3 - Y(NPW)]\tan\theta + X(NPW]$

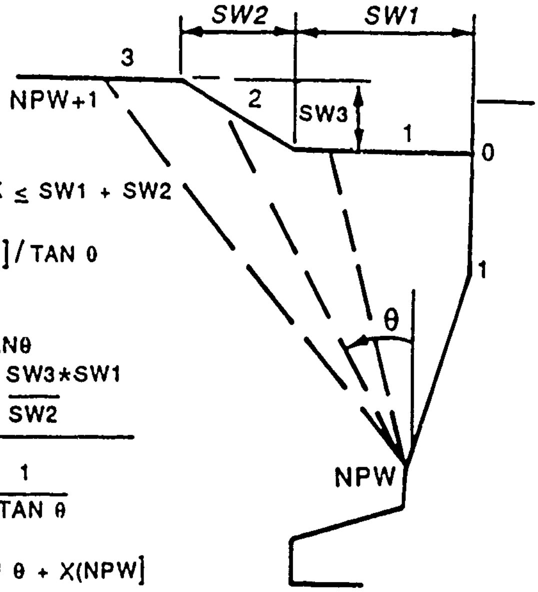


Figure 21. Intersection of wedge line/groundline

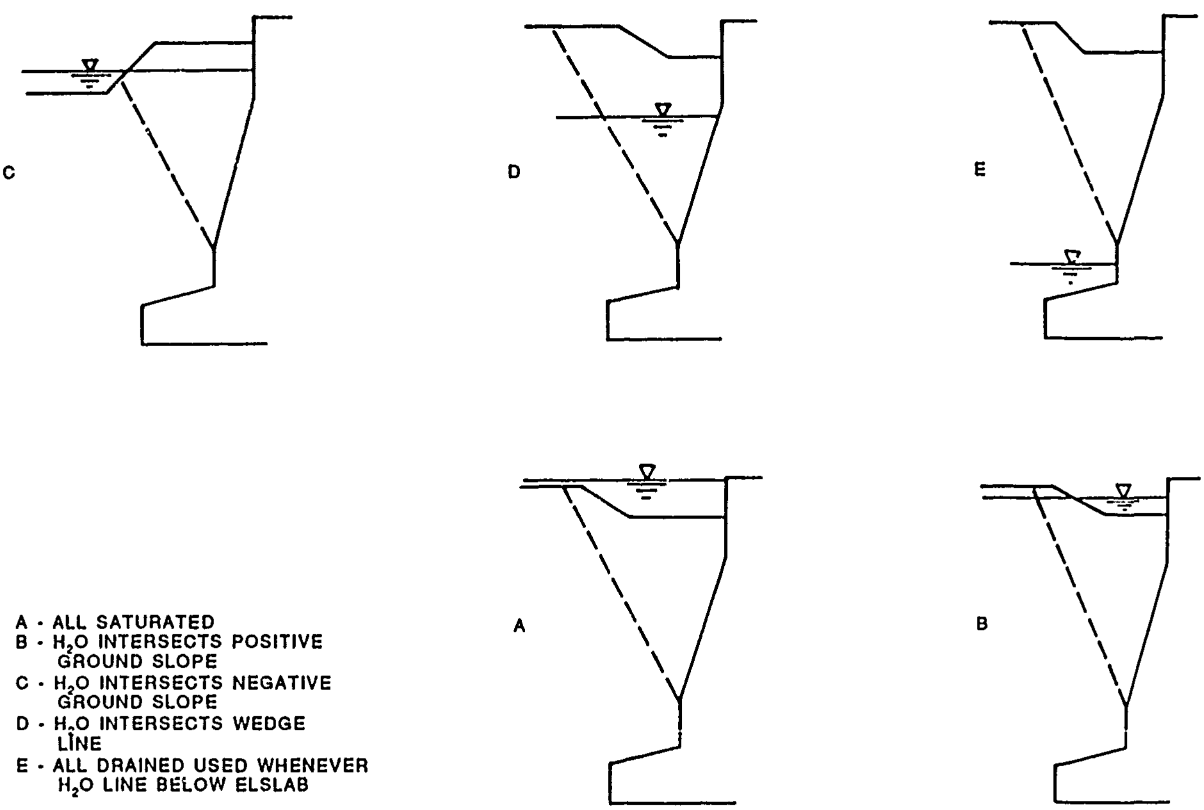


Figure 22. Combinations of dry/saturated soil

the volume of the saturated soil. The total effective weight is the sum of the drained unit weight times the drained volume plus the saturated unit weight minus the unit weight of water times the saturated volume.

80. In order to account for cracking of cohesive soils, with a cohesion value c, the following procedure is used. After the unknown force PUN(I) is found for each incremental wedge, it is tested to see if it is positive (compression against the wall). If the force is negative, it is set equal to zero and the next incremental wedge below is solved. This procedure gives a location of the depth of the crack along the wall , hcr, as approximately one-half that given by the formula

$$hcr = 2c/w*\tan (45 + \phi/2)$$

The program does not apply any hydraulic forces for water which might accumulate in the crack. However, the user may specify appropriate forces as special loads.

81. The forces from the wedge solution are used in the frame analysis module. However, for output purposes they are converted to an approximate pressure by dividing by the length of the wall or heel surface over which they act. The wedge solution was tested by verifying against a number of standard cases. For the cases where the simplifying assumptions were satisfied, the pressure distributions were in good agreement. Also, the wedge solution was tested against other wedge solutions where applicable. Again the agreement was quite good.

82. The active wedge solution is used for both exterior walls and for computing the active forces due to fill for the interior walls of a four-wall basin. At-rest forces may be approximated by specifying an appropriate at-rest factor. This factor is multiplied by the horizontal forces from the active wedge solution. If the at-rest factor is specified as one, then the forces obtained will correspond to the active case.

83. Figure 14 shows that the exterior rock elevations are input items. These input elevations are considered in the wedge solutions. The wedge solutions start as usual and proceed down the wall. However, the last incremental wedge solution is made with the bottom of the wedge taken at the top of the rock elevation. The height of this last incremental wedge will likely be less than that normally occurring. For instance, if the top of the rock occurs

somewhere above the invert elevation, the normal incremental wedge height is one-tenth of the wall height. The correct incremental wedge height is used. However, as a program expedient, the wedge force is assumed to act at the center of the normal incremental wedge height. This approximation will have no significant affect on the moment in the wall. For U-frames with no actual rock contact, the rock elevation should be set at or below the bottom of the base slab.

Passive Wedge Solution

84. Passive pressure is also based on a condition of limit equilibrium. However, the soil mass is assumed to be resisting the movement of the wall. Thus, the passive wedge solution is almost identical to the active one, except that the direction of the soil forces CRW and SRN*TAN(ϕ) is reversed from the direction shown in Figure 18 for the active wedge and that the angle α at which PUN(I) acts on the wedge is $\alpha w - \delta f$. Also, the minimum value of the incremental force PUN(I) is found in the iterative solution for the trial wedges.

85. The results of the passive wedge solution are not used directly. However, if the user selects an appropriate loading option, the horizontal forces from the passive wedge solution will be scaled along with the shear force on the base slab to provide horizontal equilibrium as described subsequently. The user should note that this procedure may result in forces on the wall on the passive side which are less than those for the at-rest case. Thus, for a U-frame that is only slightly unsymmetrical, it would be wise to run two separate solutions. Use the active solution for both walls for one run and the passive solution for one wal. in another run. Then the critical design values can be selected from the two analyses. Of course, this problem does not occur in the design mode since all loadings are symmetrical in the design mode. The results of the passive wedge solution are used for computing the horizontal equilibrium factor as described subsequently.

Empirical Wall Pressures

86. As an alternate to the wedge solutions previously outlined, an empirical wall pressure option is provided. In general, the wedge solution is

more accurate, and even though the hand calculations for the wedge solution may be lengthy, the computer time is not greatly increased by using the wedge procedure. However, some economy may be found if preliminary solutions are run with the empirical procedure. Also, it may be desirable to match existing solutions with the empirical procedure. The empirical procedure assumes that the groundline is horizontal as shown in Figure 23, and the horizontal pressure at a point is found by multiplying the effective vertical stress, PRESS, by an empirical factor, EKF, input by the user.

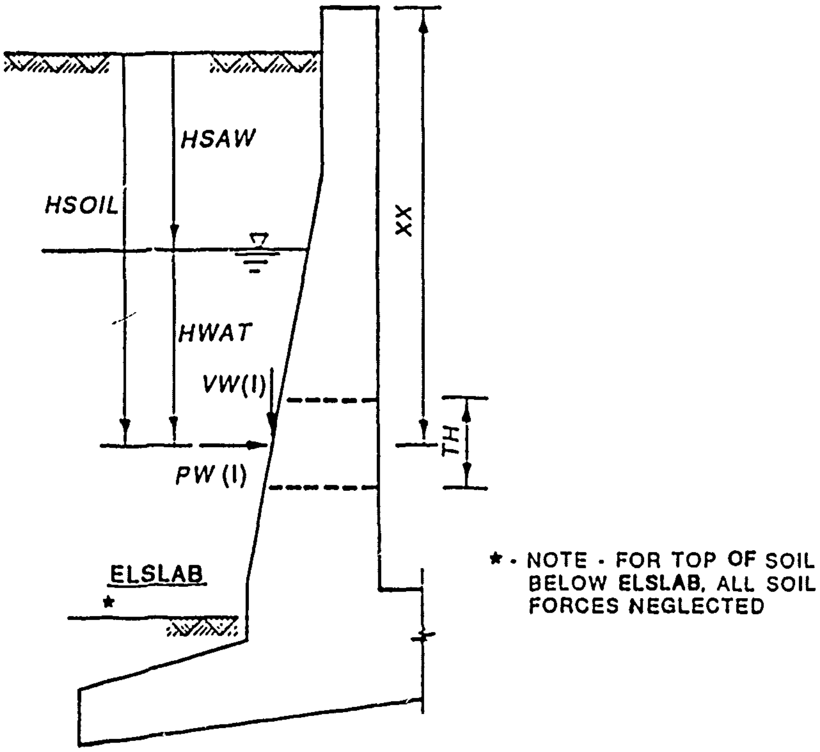


Figure 23. Empirical soil forces

87. The vertical stress is found as follows. UWD is the drained unit weight of the soil, and UWS is the saturated unit weight. GAMMAW is the unit weight of water. HSAW is the distance from the groundline down to the top of the water surface. HWAT is the height of the water above the center of the wall element considered. HSOIL is the height of the soil above the center of this element.

If HSOIL is less than or equal to HSAW, then

$$PRESS = UWD*HSOIL$$

Or if HSAW is positive, then

$$PRESS = UWD*HSAW + (UWS - GAMMAW)*HWAT$$

Or if HSAW is negative, then

$$PRESS = (UWS - GAMMAW)*HWAT + UWS*HSAW$$

88. Thus, as shown in Figure 23, the horizontal and vertical forces on the wall, PW(I) and VW(I), are computed as follows. The vertical force is simply the effective weight of the soil above the element. TH is the vertical length of the element on which the forces are being computed and is one-tenth of the wall height.
Thus,

$$VW(I) = PRESS*TH*\tan(\alpha w)$$

and

$$PW(I) = PRESS*TH*EKF$$

89. The force on the vertical face above the heel, 10 vertical and horizontal forces on the top of the heel, and the force on the vertical face on the end of the heel are computed using the same assumptions as just described for the wall. However, as in the wedge solution, if the soil elevation is below the invert elevation, then all soil forces are neglected.

90. For simple cases, the empirical solution can be made to give identical solutions with the wedge procedure and the corresponding Coulomb solution. For instance, for a U-frame with vertical walls, level backfill, and heel top, the results of the active wedge will match the empirical solution if the empirical factor is computed by the Coulomb formula

$$EKF = \tan^2(45 - \phi/2)$$

For sloping walls or heel tops, the results of the wedge solution and the empirical solution will be slightly different since the wedge solution assumes that the resultant force is normal to the surface, if no friction angle is specified.

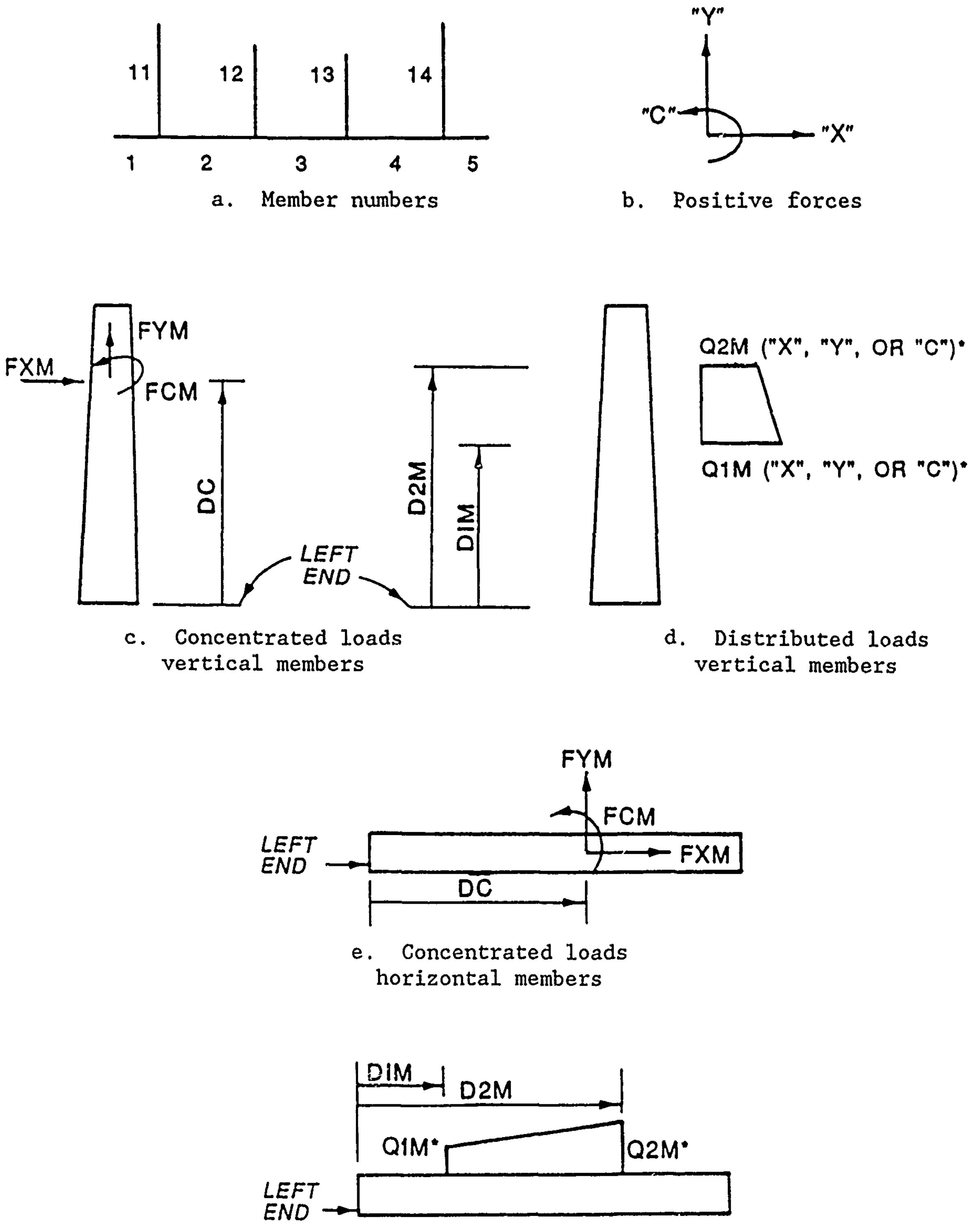
91. No at-rest factor is input for the empirical wall pressure solution. Thus, the EKF coefficient should include the at-rest correction when appropriate. Also, it will be observed by the user that the empirical factor is the same for all load cases. Thus, the user cannot adjust the horizontal forces for movement into and away from the soil as may be done with different at-rest factors for different load cases in the wedge solutions.

92. No empirical solution is given for sloping or irregular backfills. However, the user can either specify the wedge solution or estimate an approximate empirical coefficient to handle the irregular ground surface. In a manner similar to the wedge solution, no backfill force is found below the rock elevation input for the wall or heel adjacent to the rock. If there is no rock contact with the U-frame, the rock elevation should be set at or below the elevation of the bottom of the base slab.

User Specified (Special) Loads

93. The user may specify a large number of "special" distributed and concentrated loads in a simple format as illustrated in Figure 24. As described subsequently, these loads may be combined with the geohydraulic forces automatically computed if so desired. This combination feature greatly extends the capability of the program. If the users do not agree with any of the default procedures for computing the geohydraulic forces acting on the structure, they may either input the desired forces directly or add corrective forces to the ones automatically computed. In addition, forces to represent wind, earthquake, or 3-D correction forces may be applied and combined with the standard solution.

94. Since the program has nonlinear soil features, superposition of results of different load cases should not be done in general. If the special loads are combined with other loads, the loads are combined before the analysis is made. The results of two separate solutions are not superimposed. Also, the user should not try to superimpose the results of any of the load cases because of the possibility of nonlinear response and the fact that the self-weight of the frame is automatically included in each analysis.



a. Member numbers

b. Positive forces

c. Concentrated loads vertical members

d. Distributed loads vertical members

e. Concentrated loads horizontal members

f. Distributed loads horizontal members

Figure 24. Input description of special loads

95. Figure 24 shows the manner in which the special member loads are described. The member numbering sequence discussed previously is shown in the figure. All forces input are keyed to one of these members. All forces acting above the invert should be referenced to the appropriate wall member. Forces acting below the invert may be referenced to any of the members of the base slab, except missing heels, as described below.

96. It should be noted that while concentrated and distributed forces are discussed, the units of the concentrated force are kips per foot of wall and the units of distributed force are kips per foot per foot of wall or kips per square foot. Similarly, the units of concentrated couples will be kips and distributed couples kips per foot. The positive directions of all forces on either wall or slab members are shown in the figure to be to the right for horizontal forces, up for vertical forces, and counter-clockwise for couples. This coordinate system is global even though the loads are referenced to the individual members. Thus, horizontal loads are "X" loads whether they are applied to vertical or horizontal members. Similarly, "Y" loads are always vertical.

97. Forces parallel to a member are assumed applied at the centroid of the member (centroid at point of application). If the force is actually acting on a face of the member, then a couple or "C" force should also be input equal to the moment of the force about the member centroid.

98. The position of the loads are always referenced to the "left" end of the members as defined previously in Figures 4 and 8 for basins and channels, respectively. Note that the distances used for inputting special loads are referenced to the left end of the members as done to specify reinforcement locations and for output of member forces. As shown in Figure 24, concentrated loads are specified by giving the distance from the left end of the reference member to the concentrated load, DC, and the value of the concentrated load, FXM, FYM, or FCM for horizontal forces, vertical forces, and couples, respectively.

99. For convenience, any load below the base slab may be referenced to any of the slab members. Thus, the user could reference all of the loads to member one if the left heel is present. Then the horizontal distance locating all loads can be specified for the left end of member one, which is the left end of the U-frame base slab. Internally, the program will compute the proper horizontal distances to locate the forces within the proper members. However,

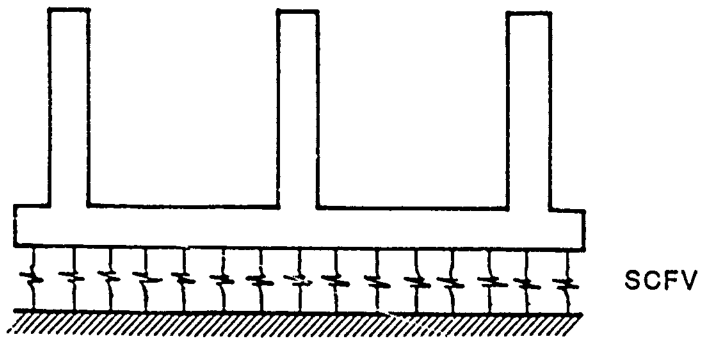
if a heel is absent, slab loads may not be referenced to the missing member. It should be remembered that the numbering of the members in the base slab is the same whether or not the heels are present. Thus, the first slab member will be member two when the left heel is omitted.

100. Distributed forces are specified by describing them as "X" forces, "Y" forces, or couples "C." Then the distances to the beginning and end of the distributed forces D1M and D2M are specified and measured from the left end of the member. Next, the values of the distributed forces at the start and end points Q1M and Q2M are input. Since all slab loads may be referenced to a single member, a linearly varying distributed load extending the entire width of the foundation may be specified as a single distributed load, with the user giving the distance to the start of the loading and the end of the loading for the chosen reference member.

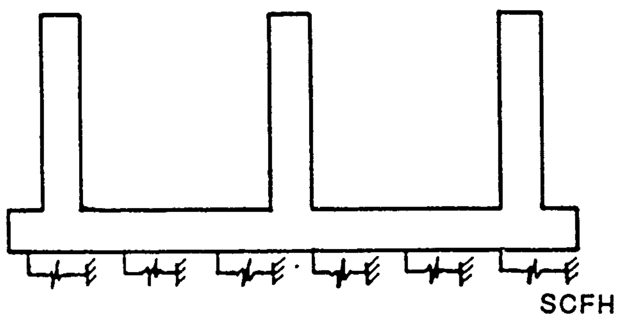
Winkler Spring Foundation

101. The Winkler assumption that the soil beneath the base acts as a series of independent elastic springs is normally used in a beam on an elastic foundation analysis. Figure 25 shows that the base is assumed to be supported by a Winkler foundation of compression only springs with a constant stiffness or spring constant SCFV. The units of SCFV are pressure per unit of deflection (kips per square inch or kips per cubic inch). The choice of SCFV can have a significant, although usually not dominating, effect on the distribution of internal forces in the U-frame. Thus, some care should be exercised in the selection of the appropriate spring constant. The availability of the program will facilitate the bracketing of significant design variables by varying the input value of SCFV.

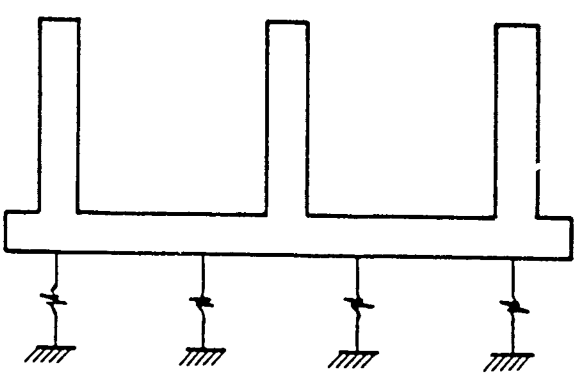
102. Distributed horizontal springs with springlike stiffness SCFH, as shown in Figure 25, are also used when the spring foundation option is selected. The horizontal shear springs are applied at the base of the slab and have the units of kips per cubic inch. The use of horizontal shear springs is not as common as vertical compression springs. However, it is important to note that for symmetrical cases the value of shear spring chosen has only a very minimal effect on the distribution of forces in the U-frame. It primarily affects the distribution of axial force in the base slab, and even this affect on the axial forces is quite small. It should be noted that for the



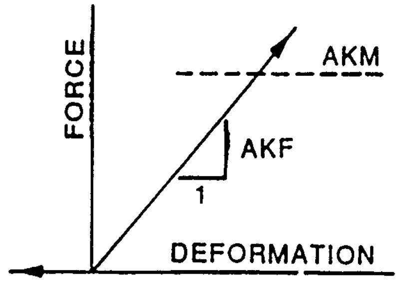
a. Vertical compression only spring foundation



b. Horizontal shear spring foundation



c. Vertical tension only anchor springs



d. Anchor response

Figure 25. Foundation spring restraints

spring foundation option the only thing providing lateral stability in the frame analysis is the stiffness of the horizontal shear springs, unless the force-deformation solution is being used for the walls. Thus, some positive value of shear spring stiffness is required.

103. A problem can also arise if the user specifies a horizontal spring stiffness that is very large compared to the other stiffnesses. This problem is due to the horizontal spring term showing up off the diagonal of the stiffness matrix since it is eccentric to the member axis. Thus, for instance, a rotation of the base slab produces a lateral movement of the bottom of the slab and a corresponding lateral force on the bottom of the slab. In the absence of detailed recommendations on horizontal shear stiffnesses, they should be taken on the order of magnitude of the vertical compression springs. The user will find that major changes in the actual input value will have a minimal change in the solution for symmetrical loadings. For unsymmetrical loads put in equilibrium with the load-deformation method for wall loading, the value of base shear spring stiffness has a more pronounced effect since it interacts with the stiffness of the springlike wall forces in providing horizontal equilibrium.

104. The vertical and horizontal base springs are assumed to be interdependent. Thus, if there is any uplift at a point along the foundation, and the compression only spring no longer provides any hold-down force, the shear spring at that location is also assumed ineffective. If uplift is a problem, then vertical anchors can be modeled as tension only springs with spring constants as shown in Figure 25. The units of the anchor spring stiffnesses, AKP, are kips per foot of U-frame per foot of deflection. The locations of the anchors are specified as described earlier in the geometry sketches for the particular basin or channel under consideration.

105. A maximum spring force, AKM, in kips per foot of U-frame is also input. However, it is important to note that as shown in the force-deformation response curve of Figure 25d, the program may compute a force that exceeds this value, i.e., elastic-plastic response is not modeled in the program. The input anchor spring maximum force is used only in computing the factor of safety for the spring and the factor of safety against uplift. The factor of safety for the spring is computed by dividing the force found in the spring into the input maximum force. Thus, a number less than one means that the anchor could not provide the force indicated by the analysis. It was

decided to program the spring response this way for two reasons. First, if the elastic-plastic response was programmed, another source would be added for instability in the analysis. Second, the real post-elastic response of the anchors is not known.

106. The fact that the base shear springs are assumed to be ineffective at points where the foundation has lost contact means that if vertical anchors are used, the U-frame would lose lateral stability if contact is lost along the entire width of the base slab. In reality, some lateral stability would be provided by the force-deformation response of the soil against the sides of the U-frames. It is probably best to use a force-deformation solution for the walls for such cases. However, if the loading is close to symmetrical, it is acceptable to simply artificially stabilize the U-frame with fictitious lateral springs of small stiffness. The fictitious lateral springs are automatically provided for in the program whenever the user specifies vertical anchors.

107. In spite of the generally highly nonlinear response of the frame when uplift is a problem, the solutions generally converge with little difficulty. The few cases where convergence has not occurred were generally associated with excessive uplift and having only a minimal number of anchors effective in resisting uplift.

Empirical Foundation Pressures

108. The active loads may be put in equilibrium by an empirical foundation procedure rather than by the Winkler spring foundation model just described. The Winkler spring foundation is considered the more rational approach. However, some small economy in computer time may be obtained in using the empirical procedure, and the empirical approach may be convenient for matching existing design calculations.

109. Figure 26 illustrates the empirical procedure for satisfying vertical and rotational equilibrium. SUMFY is the sum of all active vertical forces, and SUMM is the resultant moment of all active forces about the center of the base slab at the bottom of the slab. The empirical procedure is based on a "P/A + Mc/I" approach except that the "P/A" distribution may be nonuniform. The dashed line distribution in Figure 26 shows the assumed distribution if the sum of the moments, SUMM, was zero. The user specifies the ratio,

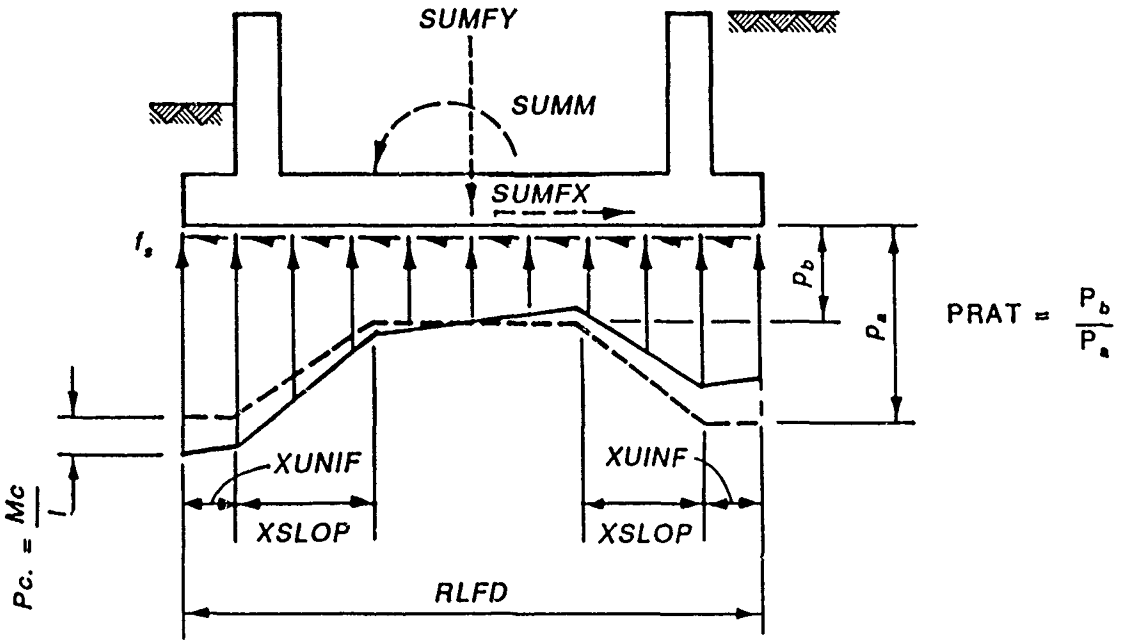


Figure 26. Empirical foundation pressures

PRAT, of the inner pressure Pb to the outer pressure Pa. Input of the distances XUNIF and XSLOP as defined in the figure are also required.

110. Then the pressure Pa is computed such that the dashed line pressure distribution will put the force SUMFY in equilibrium, as follows:

Pa = SUMFY/[(1-PRAT)*(2*XUNIF+XSLOP) + PRAT*RLFD]
Pb = PRAT*Pa

Then based on rotational equilibrium and assuming a rigid foundation, the additional pressure Pc due to the moment is found as

$$Pc = 6*SUMM/(RLFD^2)$$

The total pressure at any point is easily found by summing the pressure from the "P/A" and "Mc/I" solutions.

111. The foregoing solution was developed assuming contact between the soil and the U-frame across the full width of the foundation. If contact is lost, an incorrect tension (negative) foundation pressure will be calculated and the program will output a warning message. It would be possible to

develop an empirical solution for the case where contact is lost. However, this step was not taken since the elastic spring foundation procedure should be used for such cases. The resultant horizontal force, SUMFX, is put in equilibrium by the uniformly distributed pressure, fs, across the bottom of the slab.

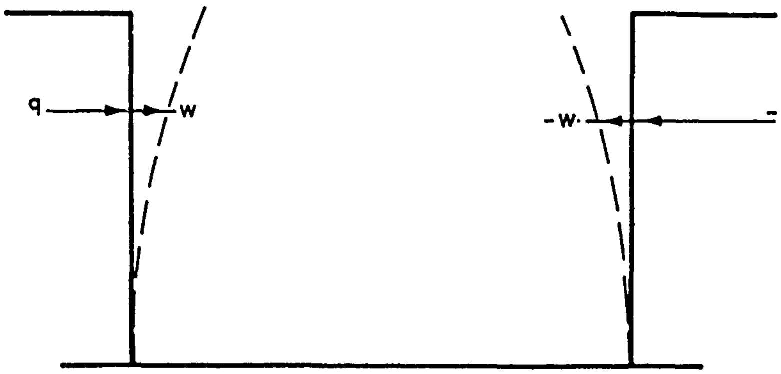
112. When the empirical foundation option is used, then the total forces applied to the U-frame module will be in equilibrium prior to going to the frame solution. However, rigid body restraints must be provided to allow the frame solution to proceed. Rigid body motion is prevented by one horizontal and two vertical springs. While these springs develop no force and do not affect the distribution of internal forces in the U-frame, they do prevent rigid body motion in an arbitrary manner. Thus, the deflections computed in the frame module are meaningless and are not output for the empirical foundation option.

Load-Deformation Solution for Wall Loading

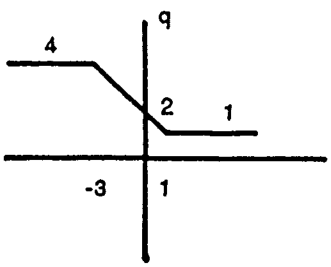
113. The active and passive states of soil pressure are limit states of the more general nonlinear load-deformation response of soil to the motion of the wall. If the wall moves sufficiently into the soil, an upper limit of passive pressure is reached. When the wall moves far enough away from the soil, a lower limit of active pressure is reached. In between these states the soil pressure acting on the wall is a nonlinear function of the displacement of the wall. The exact nonlinear relationship is quite complex and depends on the soil parameters, the wall friction, and the construction technique.

114. Haliburton (1972) has given rules for a simple elastic-plastic relationship between the active and passive states. More detailed studies are needed with correlations with testing and rigorous finite element solutions to develop force-deformation relationships that are precise. Meanwhile, the program can be used to aid in such studies and to allow the designer to see the effect of the interaction of wall deflection and soil pressure on the forces developed in a U-frame structure.

115. Force-deformation curves are described as q-w curves herein. The general nature of the curves for a symmetrical U-frame is illustrated in Figure 27. The curves shown in the figure are of the elastic-plastic type.

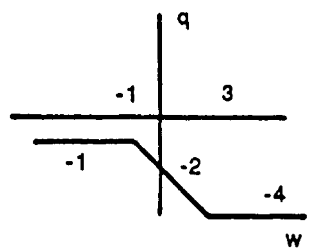


a. U-frame symmetrically loaded



W	-	-3	0	1	+
q	4	4	2	1	1

b. Left wall curve



W	-	-1	0	3	+
q	-1	-1	-2	-4	-4

c. Right wall curve

Figure 27. Input description of force-deformation curves

However, the curves may be input by a series of up eight points. The units of q are pressure (kips per square foot), and the displacements are in feet. Positive pressure and displacement are to the right. Thus, the signs of curves for the left and the right wall will be reversed as shown in the figure. Also, the order the points are input will be reversed. The program allows, however, for the description of these symmetrical and reversed curves through the input of a negative curve number. If the curve number input is negative, then the values used for the negative curve are obtained by reversing the order of the input points and changing the signs of the curve with the same absolute value as the negative curve number. Also, the curves may be scaled by giving basic curves and then multipliers of the basic curves at different locations along the walls.

116. Curves may be used to represent soil or rock force-deformation response on any of the walls of the U-frame. However, the rock elevations shown in Figure 14 are not input for the force-deformation option. The user must specify appropriate q-w curves at various elevations to model the soil and/or rock stiffnesses.

117. The force-deformation response is only in the horizontal direction for the walls. Thus, no vertical forces are developed on the wall and no forces are developed below the bottom of the wall members (the invert elevation). Any vertical wall forces or active soil forces on the heel must be input as special forces. Of course, the reactive forces on the base slab and heel will be obtained from the spring foundation solution.

Program Loading Combinations

118. The various program options for active and reactive loadings have already been described. In this section, the ways in which they may be combined are described. Section 7 in the input guides (contained in the user's guides, Volumes B and C) is the loading control section. Here the user specifies the following control parameters. As mentioned earlier, there are certain restrictions on the loading for the design mode which will be discussed later.

119. NEM is the number of "EM-like" load cases (1-10). These load cases are governed by water and fill elevations using the various options described earlier. However, if the load-deformation solution is used for the

wall loading, then fill elevations are not used and the program has the following restrictions. For load-deformation solutions, only one EM-like load case is permitted and there must be one special load case (NSPEC = 1). All active loads (U-frame weight, hydraulic loads, and special loads) are combined before the frame analysis is made; the frame analysis puts these loads in equilibrium with the wall loading generated by the force-deformation curves and the foundation reaction pressure developed using the spring foundation option.

120. NSPEC is the number of special load cases (1-3). These load cases are specific loadings described with the various members of the frame being considered. However, except when using the load-deformation solution for lateral wall pressures, the user may combine the special load cases with any one of the previously defined EM-like load cases, if desired, by giving the reference number of the EM-like load case.

121. For instance, suppose three EM-like load cases are run followed by two special load cases, and the first special load case references the third EM-like load case while the second special load case does not reference an EM-like load case. The fourth load case would be for the combined active loads of the third EM-like load case and special load case one. The fifth load case would be for the active loading of special load case two only plus the self-weight of the U-frame. All load cases have reactive loadings computed with the options exercised and automatically include the weight of the U-frame using the input concrete unit weight.

122. BTYPE is the type of analysis for the backfill, including divider fill if present. For BTYPE = "WEDA," the backfill pressure is computed using active wedge solutions for all walls with backfill. For BTYPE = "WEDPL," a passive solution is made for the left wall, and active solutions are made for all other walls with backfill. For BTYPE = "WEDPR," a passive solution is made for the right wall, and active solutions are made for all other walls with backfill. When a passive solution is made for either wall, it is adjusted to provide the equilibrium of all horizontal forces in conjunction with the horizontal base shear as described subsequently.

123. For all active wedge solutions, the at-rest factor will be multiplied times the value of horizontal forces and pressures originally obtained. Thus, if no at-rest correction is desired, then the at-rest factor should be specified as 1.0. For BTYPE = "EMP," the backfill pressure is computed using

the empirical procedure previously described. For BTYPE = "LDM," a load-deformation solution is made for the horizontal loading on the walls.

124. FTYPE is the type of foundation analysis used to compute the reactive loading to provide equilibrium. For FTYPE = "EMP," the active loads are put in equilibrium through the empirical procedure previously described. For FTYPE = "SPR," the active loads are put in equilibrium using the beam on elastic foundation procedure.

125. It is important that the user understand the significance of how these loading options interact. Figure 28 shows a schematic flowchart of the solution procedure with particular emphasis on the interaction of the active and reactive loadings in the various options. The flowchart illustrates the general case in which a special load is combined with an EM-like load case. For an EM-like load case alone, the special loads would be omitted. For a special load case alone, only the special loads and concrete weight would be acting prior to the foundation solution.

126. As shown on the flowchart, first the self-weight is combined with the hydraulic and special loads. Then depending on the backfill option, additional active loads may be computed. For an empirical or active backfill solution (BTYPE = "EMP" or "WEDA"), the appropriate active wall and heel loads are found and combined with the previous active loads before going on to the foundation solution. Then during the foundation solution, either empirical or Winkler, the equilibrium is established. If there is any unsymmetrical horizontal loading on the U-frame, then it will be balanced only by the shear force on the foundation for these foundation options.

Horizontal Equilibrium Factor

127. For BTYPE = "EMP" or "WEDA," a horizontal equilibrium factor, HEF, is computed as illustrated in Figure 29. The 20-kip foundation force shown is the maximum shear capacity of the base computed by multiplying the input cohesive stress times the full width of the base slab and adding the product of the resultant vertical force on the base slab (if upwards) times the tangent of the input base friction angle. The base shear force required for equilibrium is 5 kips as shown in the figure. Thus, the horizontal equilibrium factor is four. If the horizontal equilibrium factor is less than one, the solution may still proceed at the discretion of the user. However, if the

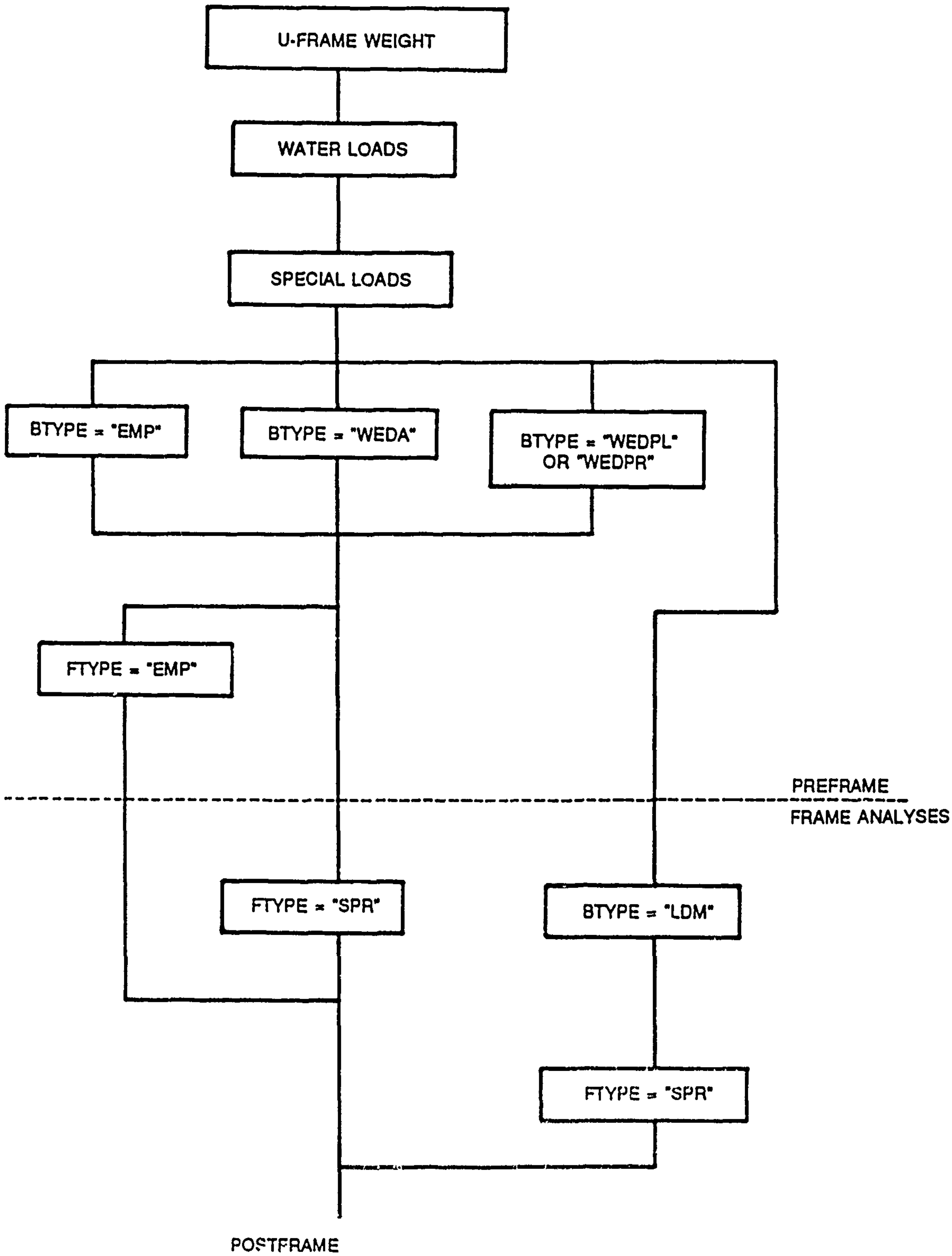
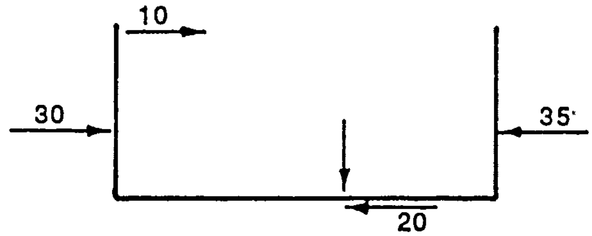


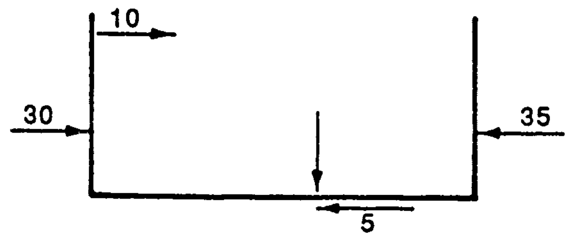
Figure 28. Schematic flowchart of load options



a. Active forces/maximum base shear

$$HEF = \frac{20}{30 + 10 - 35}$$

$$HEF = 4$$

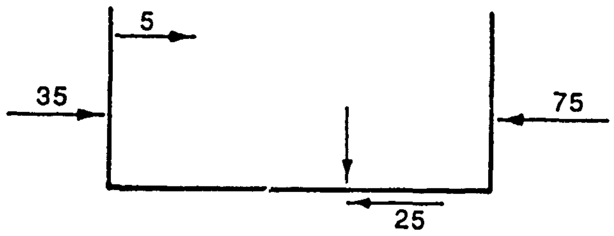


b. Active forces/equilibrium base shear

Figure 29. Horizontal equilibrium for BTYPE = "WEDA"

solution continues, then the computer will be using a base shear larger than the maximum capacity computed for the foundation.

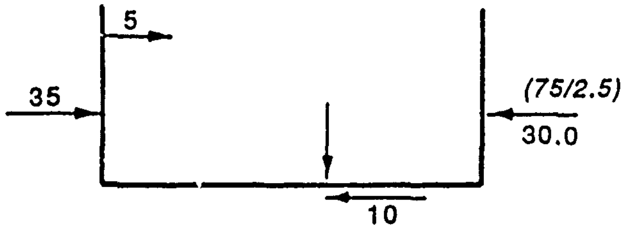
128. If a passive solution is specified for either the left or the right wall, then the appropriate passive solution is accomplished with an active solution made for all other walls. Then the horizontal equilibrium factor is computed as shown in Figure 30. Again, the maximum capacity of the base shear is computed and now added to the full passive wall force in computing the horizontal equilibrium factor as illustrated by the example in the figure. Then the passive wall force is divided by the horizontal equilibrium factor to yield the wall force acting on the passive side under equilibrium conditions. The base shear force is then actually found in the solution of the base for equilibrium (either the empirical or spring foundation solution as shown in Figure 28). However, the result will always be the same value as simply dividing the maximum base shear possible by the horizontal equilibrium factor. As for the empirical and active backfill options, the solution should



a. Active forces/maximum passive forces

$$HEF = \frac{(75 + 25)}{(35 + 5)}$$

$$HEF = 2.5$$



b. Active forces/equilibrium passive forces

Figure 30. Horizontal equilibrium for BTYPE = "WEDPR"

be allowed to continue only if an adequate horizontal equilibrium factor is obtained.

129. If any portion of the base slab uplifts, then the portion of the maximum horizontal force computed for the base slab will be in error, since the entire width of the base slab was multiplied times the maximum foundation cohesion. No correction was made in the program for this uplift because the amount of contact at the time of potential sliding is not known. If the elastic foundation module is used, the locations at uplift under the nominal loading would be known. However, the uplift may be different under conditions in which the maximum foundation force would be acting. Thus, in cases where uplift occurs or is impending, the value of cohesion input for the base slab should be a conservative value.

130. Figure 28 shows that if the load-deformation solution is used for the wall loads, then equilibrium must be provided by spring foundation reactions only. Thus, the empirical option cannot be used for the foundation in

conjunction with the load-deformation solution for the walls. This restriction is necessary because the load-deformation wall solution is performed as an iterative solution as part of the overall frame analysis. Since the load-deformation solution is an equilibrium solution based on compatible displacements, no horizontal equilibrium factor is computed for the load-deformation solution.

131. It will be noted by the user familiar with sliding stability calculations that the horizontal equilibrium factor is somewhat like the factor of safety with respect to sliding. However, the procedure used is not the same as and will yield values different from those found using the procedure outlined in ETL-1110-2-256, "Sliding Stability For Concrete Structure (Headquarters, Department of the Army 1981). The primary purpose of the U-frame program is to find the forces acting on the walls under the design loading condition. If the sliding stability is in question, then a separate sliding stability analysis should be made.

Uplift Factor of Safety

132. The factor of safety against uplift, FSUP, is computed as follows. WUF is the weight of the U-frame, and WSOIL is the sum of all the vertical components of the soil forces acting on the U-frame. WSPEC is the sum of all the vertical components of the special forces acting on the U-frame. FHOLD is the sum of the maximum anchor forces input for all anchors, and WWATI is the sum of the weight of all the water contained within the U-frame. All of these forces react against the total uplift force UWAT to provide stability. UWAT is the algebraic sum of the uplift forces on the bottom of the base slab and the weight of the water on the external walls and heel. Thus,

$$FSUP = (WSOIL + WUF + WSPEC + WWATI + FHOLD) / UWAT$$

A factor of safety against uplift is computed for all load options except for that of special loads only since, for that case, there would be no hydraulic forces specified.

133. If a factor of safety against uplift less than 1.0 is obtained, equilibrium cannot be maintained within the conditions specified by the data and generally the problem should be terminated. However, the program does

allow the user to continue, because for the foundation with anchors a solution would still be possible. However, one or more of the anchors would have forces in excess of the input maximum values. If the spring foundation is used and there are no anchors present, then equilibrium is not possible for an uplift factor of safety less than 1.0. In fact, numerical problems may occur if the factor of safety against uplift is less than about 1.01.

134. For the empirical foundation solution, a nonsensical solution involving tension between the base slab and the soil would be obtained for a case with an uplift factor of safety less than 1.0. If the user allows such a solution to proceed, then a warning message will be included in the output.

PART V: RESULTS OF ANALYSIS

General Description of Program Output Options

135. The program allows a variety of output options involving partial, detailed, and graphical output. A complete listing of the input data, with appropriate headings, will be generated with the output file. For the design mode, original and final values are shown for the design variables. Detailed examples with input and output are found in Volumes B and C. Also, a sketch of the frame geometry, water elevations, and ground profile, as shown in Figure 31, may be obtained. The figure shows a three-basin structure with two heels and both wall and slab drains. Note that the member numbers used in describing the member loads, reinforcing, and output are shown on the sketch. The ground profile and rock elevations are plotted, and the water elevations are shown for the EM-like load cases.

136. For the investigation mode, no pass-fail decisions are made by the program; all results are presented, and the user makes the decision of the adequacy of the structure. For example, if the SD option is used, the strength and ductility ratios are computed and output at the various sections requested by the user. However, no messages are printed if these values exceed 1.0. Further, no strength checks are made at any section not requested by the user.

137. In the design mode, either the section selected satisfies all the criteria checked by the program, or appropriate warning messages will be issued. The user should review the output for such messages, as well as the complete output and the assumptions and limitations of the program, before accepting the results of the program as an acceptable design.

138. The remainder of this part of the report is devoted to the output for the investigation mode. Much of this output is also available in the design mode. Part VI of this report describes in detail the design mode and the special output for the design mode.

Factors of Safety

139. The factor of safety concerning uplift is computed as described earlier. The factor of safety against excessive bearing pressure is computed

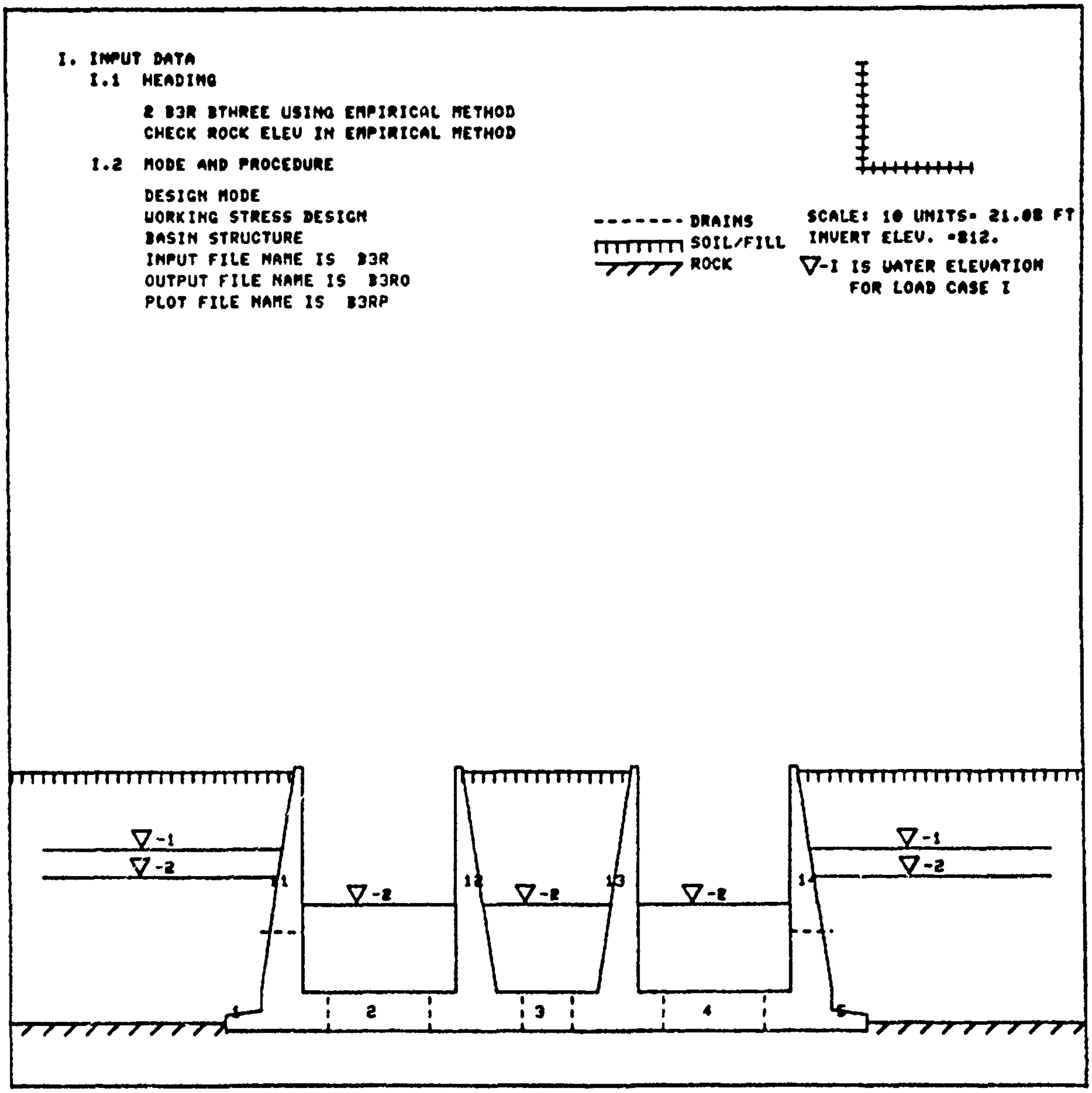


Figure 31. Geometry plot for three-basin U-frame

by dividing the maximum foundation pressure developed in either the empirical or the spring foundation option into the maximum foundation pressure specified for the foundation. The horizontal equilibrium factor described earlier is output with the factors of safety concerning uplift and bearing. However, it should not be considered to be a factor of safety in sliding according to ETL-1110-2-256 (Headquarters, Department of the Army 1981).

140. Depending on the loading options exercised, some of the above factors may not be known prior to the frame analysis solution. Generally, the

program will output the factors, and the user has the option of stopping the analysis before going to the frame solution if any of the factors are not satisfactory. For the load-deformation solution, no horizontal equilibrium factor is computed.

Output of Member Pressures

141. Output of pressures along the faces of the U-frame are organized in terms of the members used for describing the frame. The signs used for all pressures are the same as that used for loads; horizontal pressures are positive to the right, and vertical pressures are positive if up. All of these directions refer to the direction of the pressure on the U-frame, regardless of the member or face on which the pressure acts. Thus, the horizontal water pressure shown on the right of the wall in Figure 32 would be negative.

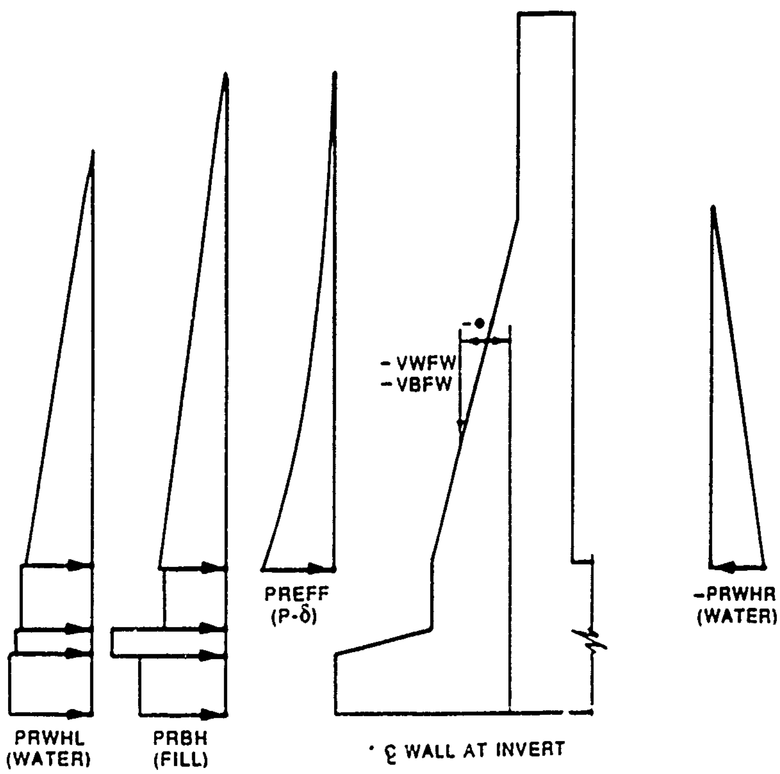


Figure 32. Pressure output for walls

142. Figure 32 shows the type of pressure output available for a wall. Wall pressures are computed and output at 11 equally spaced points from top of the wall to the invert. The pressure is computed by first taking the

corresponding force acting at the middle of the 10 equal elements along the wall used in the frame solution and dividing by the vertical length of the element. This computation gives the approximate pressure at the middle of the elements. Then the pressures at the nodes at the ends of the elements are obtained by interpolation for the interior nodes and by extrapolation for the end nodes. This procedure may sometimes give a slight pressure with the wrong sign at a node close to the point of zero pressure. As a result, the program will output a zero pressure at that node. However, it should be remembered that these pressures are computed only for convenience in the output. The correct forces were used in the frame solution.

143. The output pressures available for the wall members are as follows:

- a. PRBH is the horizontal component of the backfill pressure.
- b. PRWHL is the horizontal component of the water pressure acting on the left side of the wall.
- c. PRWHR is the horizontal component of the water pressure acting on the right side of the wall.
- d. PREFF is the horizontal pressure from the nonlinear force deformation solution.

The net lateral pressure which is the sum of all pressures acting on the wall is also available. However, that output is included with the member force output and will be described later.

144. For external walls, values for water pressure and backfill pressure will be also be available below the invert as shown in the figure. While the pressures are given at 11 equally spaced points above the invert, the values below the invert are only given at the centers of the three surfaces along the heel. Note that the magnitude shown for the sloping surface of the heel for PRBH is considerably higher than for the two vertical heel surfaces. This difference is due to the wedge solution which gives higher horizontal pressures on a sloping surface than along a vertical surface. A similar effect occurs in the Coulomb solution for lateral earth pressure.

145. In addition to the lateral pressures, vertical resultant forces on the wall are also output for the backfill and water, VWFW and VBFW, respectively. The signs of these resultant forces are the same as for the pressures. The units of the forces are kips per foot of wall. The eccentricities of these forces from the center of the base of the wall are also listed. The

eccentricities are positive if to the right. Thus, the vertical wall forces and eccentricities are all negative as shown in Figure 32, as would normally be the case for the leftmost wall.

146. Numerical values of these output pressures and resultant forces are placed in the output file for all wall members. Also, the horizontal components of backfill and water pressure may be plotted for the wall members as shown in Figure 33. The sample plot shows the output for an external wall of the U-frame presented in Figure 31. The direction of the pressures are shown in addition to the sign.

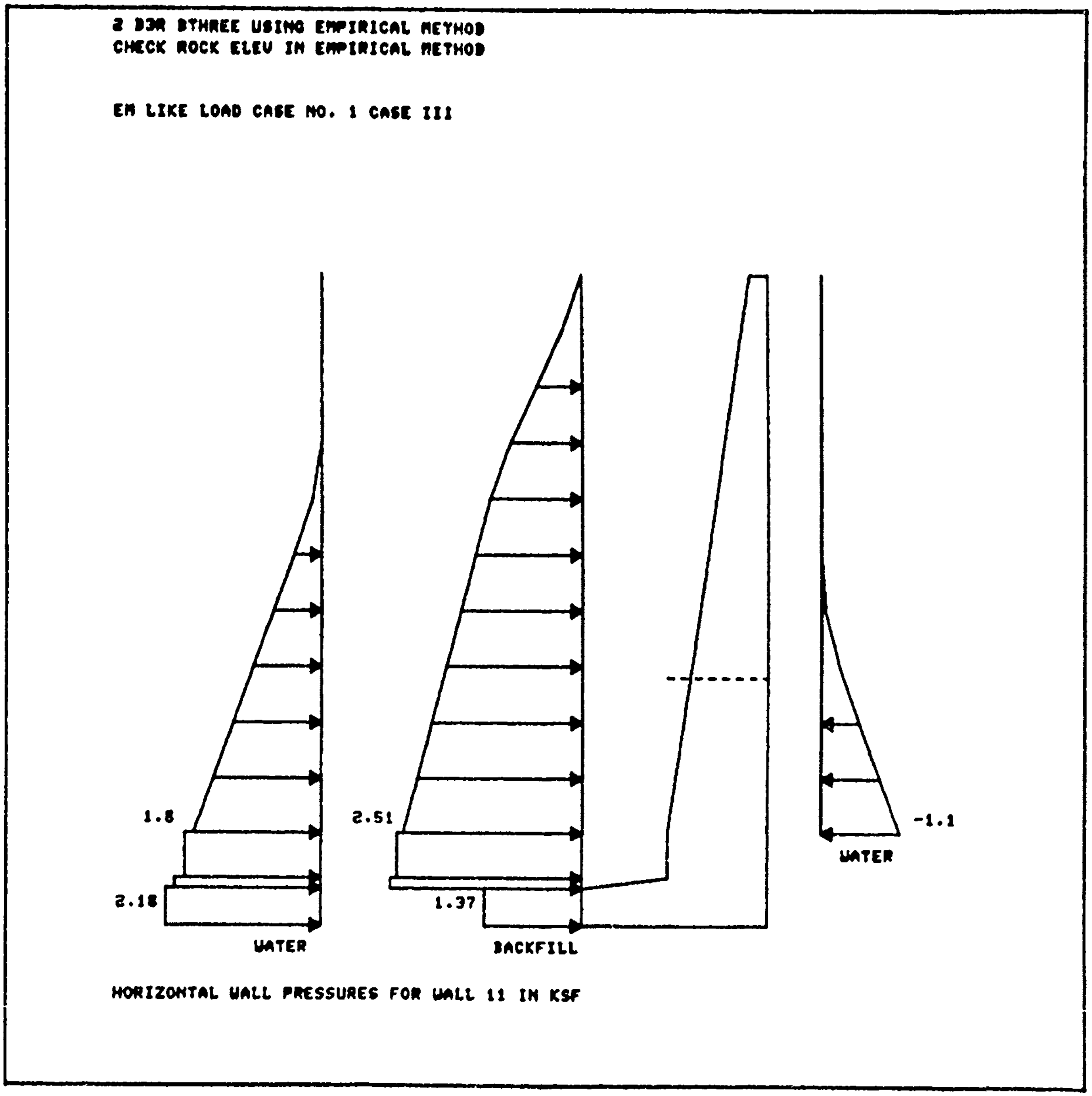


Figure 33. Sample wall pressure plot

147. Note the figure does not show any significant effect of the increased soil pressure on the sloping face as described for the previous figure. This behavior is due to the fact that the change of elevation over the sloping face of the heel is less than a foot. Also, the backfill pressure plotted for the bottom face of the heel is lower than the pressure on the heel at a higher elevation. This lower output pressure is due to the fact that the rock elevation was set along the lower vertical face of the heel as shown in Figure 31. The correct horizontal force was computed for the wedge taken with its lowest point on top of the rock surface. The pressure output is an "average" over the full height of the vertical surface of the heel.

148. Figure 34 shows the pressures and resultant forces which are stored in the output file for the members of the base slab, including the heel. The same sign convention is used as for the walls. The following pressures are available:

a. PRBV is the vertical component of the backfill pressure.

b. PRWDV is the vertical component of the water pressure on top of the slab.

c. PRWUV is the vertical component of the water pressure on the slab bottom.

d. PREFF is the vertical effective foundation pressure from either the spring or empirical foundation solution.

149. Numerical values are given at 11 equally spaced nodal points for all the interior slab members. Values are given for the heels at the ends and midpoint. Also, values are given for the rigid blocks under the walls at their ends. Pressures for output at these nodal points are computed from the forces acting at the center of the elements in a manner similar to the procedure described for the walls.

150. In addition to the pressures listed, the values of the resultant forces as shown in the figure are stored in the output file.

a. HBFH is the horizontal force from the backfill acting on the vertical face on the end of the heel.

b. HWFH is the horizontal hydraulic force acting on the same face.

c. HBFHT is the horizontal force from the backfill acting on the sloping heel surface.

d. HWFHT is the horizontal hydraulic force acting on the same face.

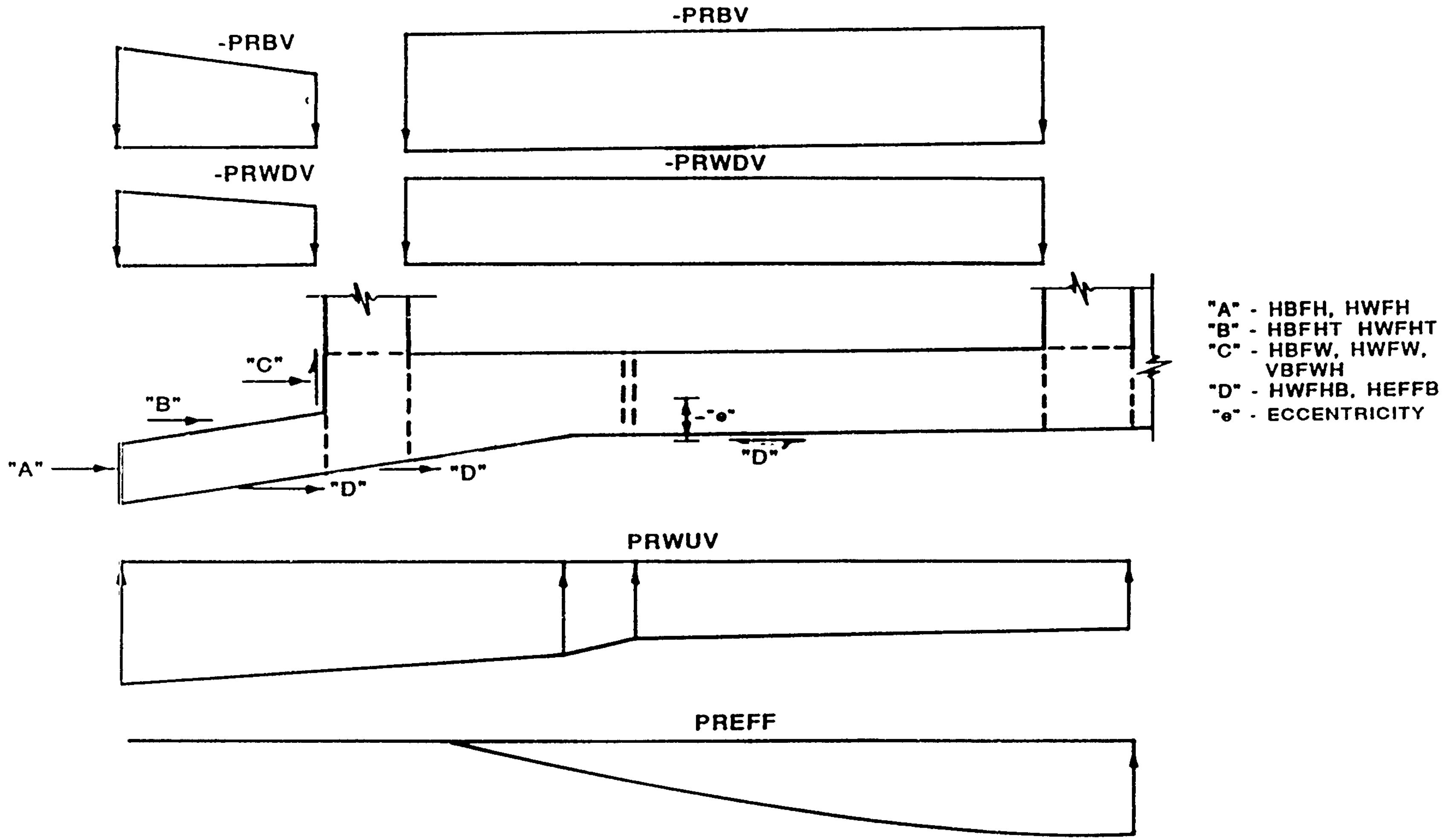


Figure 34. Pressure output for base slab

_e_. HBFW is the horizontal force from the backfill acting on the vertical face of the wall below the invert.

_f_. HWFW is the horizontal hydraulic force acting on the same face.

_g_. VBFWH is the vertical backfill force acting on the same face.

_h_. HWFHB is the horizontal hydraulic force acting on the bottom of the slab.

_i_. HEFFB is the horizontal effective foundation force acting on the bottom of the slab.

Numerical values of the above forces and their eccentricities from the centroids of the left end of the member or end block are given for the slab members and rigid blocks under the walls as indicated in the figure.

151. The vertical pressures acting on the base slab may also be plotted as shown in Figure 35. The outline of the base slab is seen with the water pressure on the top and the bottom of the slab plotted adjacent. The effective foundation pressure is seen at the bottom of the figure, while at the top of the figure the vertical component of the fill pressure is plotted.

Output of Member Forces

152. Member forces are computed in the frame analysis module at 11 equally spaced points along the vertical and horizontal members. However, for the heels, the forces are only output at both ends and the middle of the heels. These forces may be obtained in both tabular and graphical form. The force quantities available are the axial force, AXIAL, shear force, SHEAR, and bending moment, BMOM. Positive values of these forces are shown in Figure 36 for both horizontal and vertical members. The sign convention used is a designer's convention rather than a frame convention. Thus, a positive moment produces tension on the "bottom" of the member, a positive shear produces a clockwise couple on the element, and a positive axial force is in compression. The distance to the output point from the "left" end of the member, DIST, is included in the tabular output along with the thickness of the member at the output node, THICK.

153. Simultaneously with the force output, the net lateral pressure, PNETL, is output. This net lateral pressure is simply the sum of all the acting pressures and is useful for checking the equilibrium of the members. The corresponding lateral deflections of the member, LATD, are also tabulated

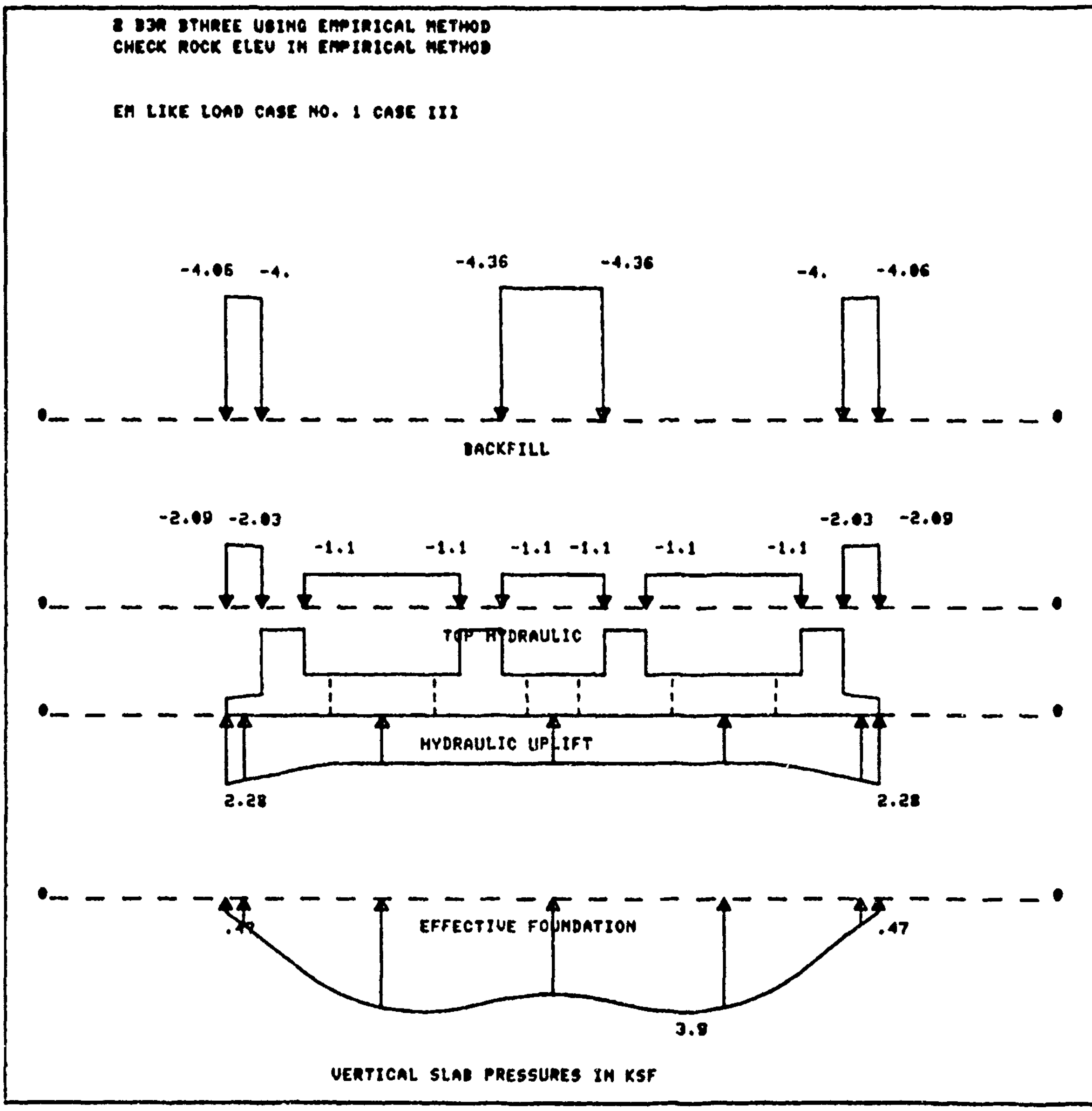
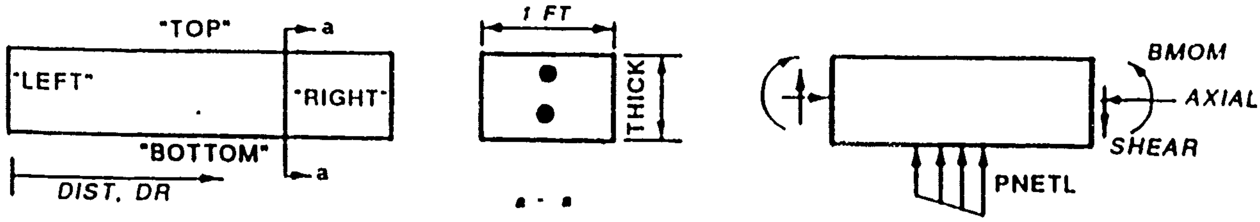
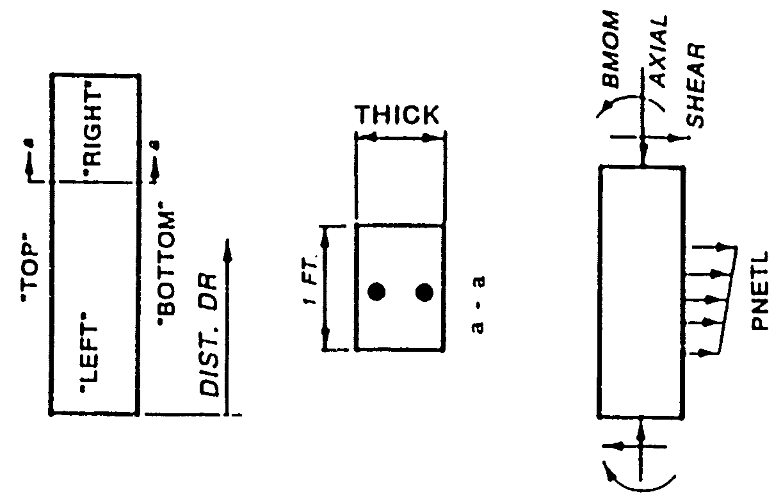


Figure 35. Sample base pressure plot



a. Member output - horizontal members



b. Member output - vertical members

Figure 36. Positive member output quantities

except for the empirical foundation option. The signs for the pressure and deflections are the same as for all the other pressures, i.e. to the right and up are positive.

154. Graphical output of all these quantities may be obtained, member by member, for each load case as illustrated by Figures 37 and 38 for a typical slab and wall member, respectively. The results for the wall show that the wall has deflected to the left because of the net pressure to the left from the divider fill. Thus, a negative shear and positive moment situation on the member whose "bottom" is at the far right was created.

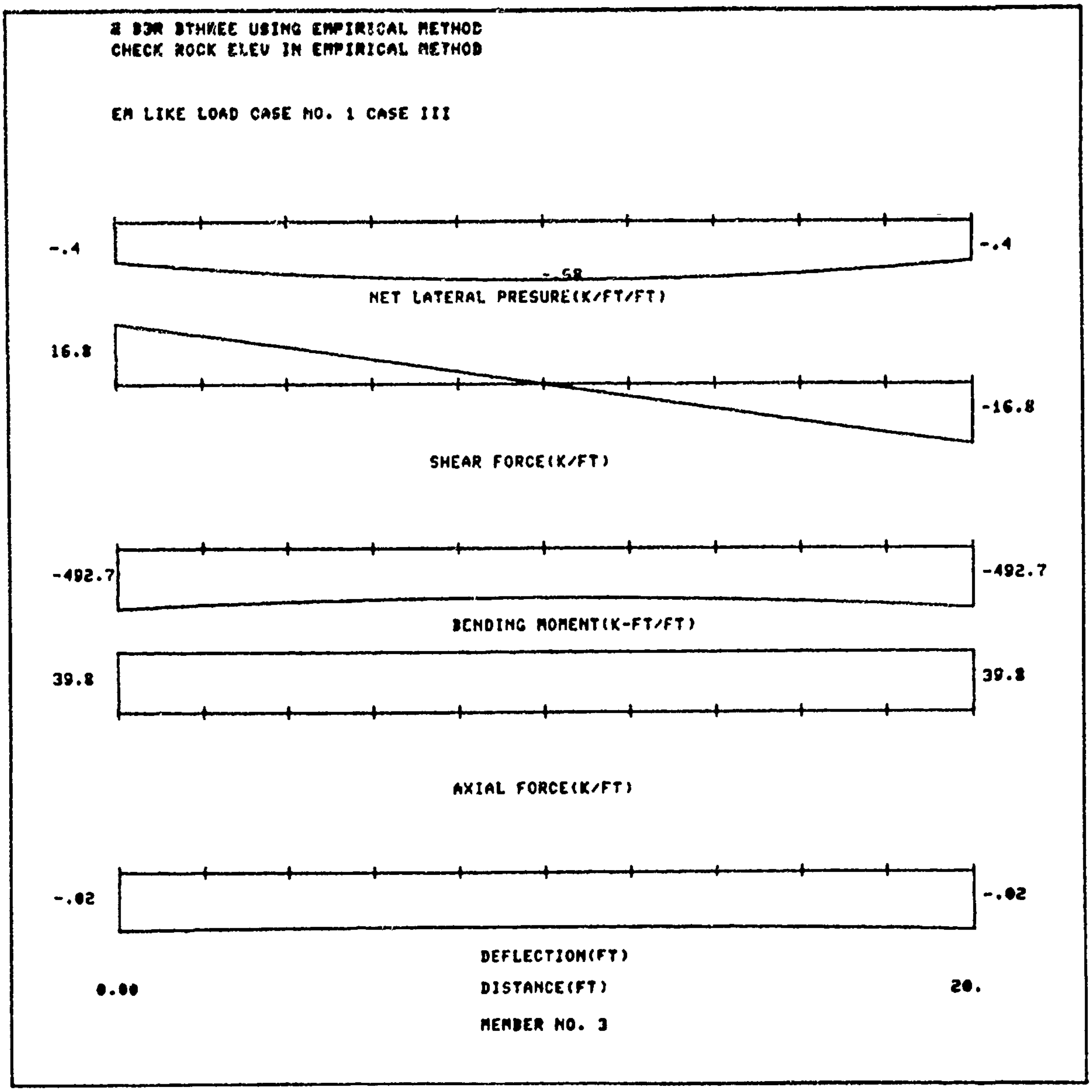


Figure 37. Sample member force plot for slab member

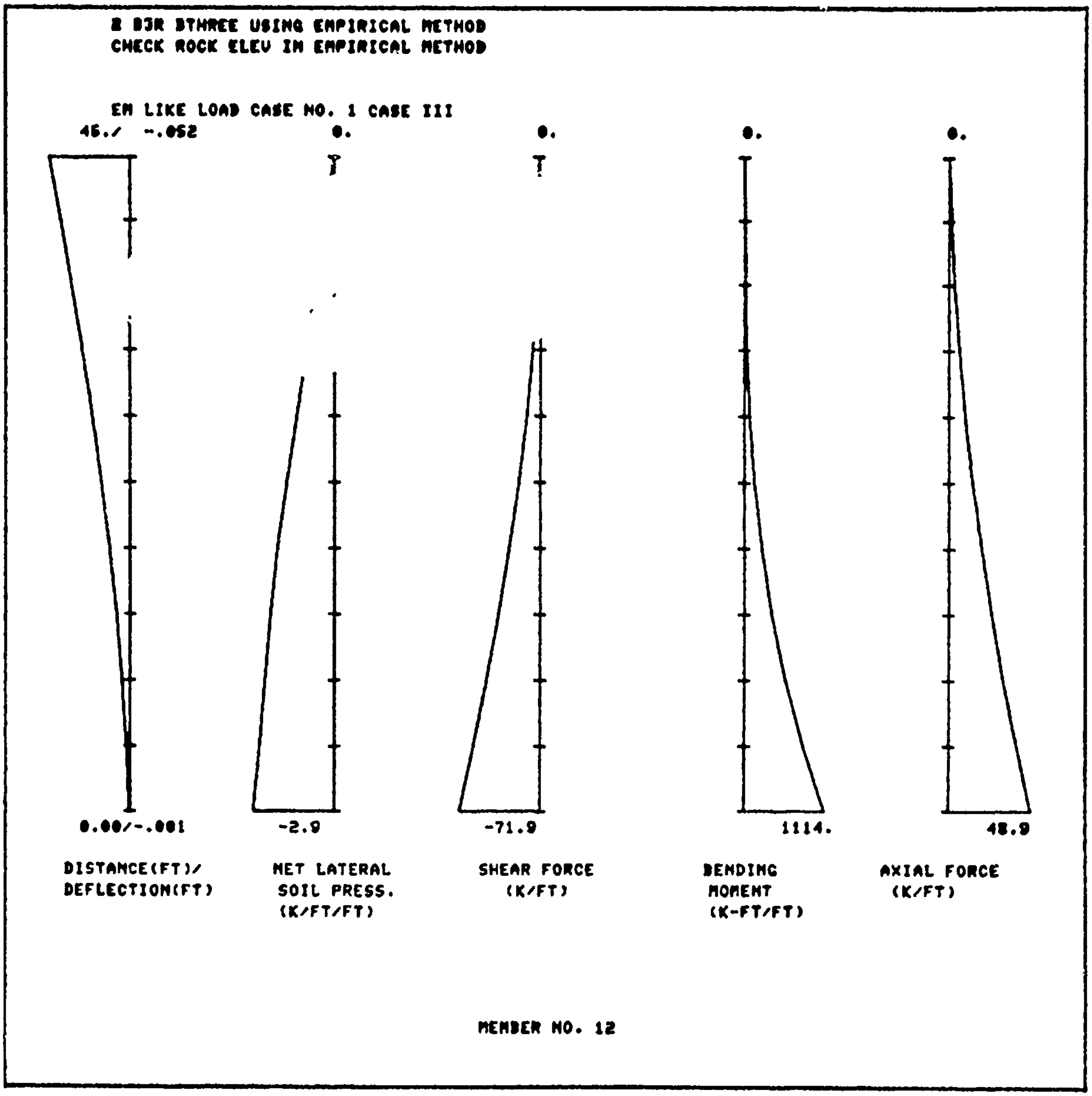


Figure 38. Sample member force plot for wall member

Output of Member Stresses in Investigation Mode - WSD Option

155. Stresses can be computed by traditional formulae associated with allowable stress design, as described subsequently, at up to five points per member. The locations of these points and the reinforcing at the locations of the points must be specified by the user as shown in Figure 5. The user specifies steel reinforcing for the sections at the "top" and "bottom" faces as previously described. A warning message is output whenever the user fails to specify steel on the "tension" side of the section. Stress output is only provided for members for which it is requested. The stress output is listed for each load case following the other member output for those members for which it is requested.

156. The axial force, shear, and moment at the section being investigated are found by linear interpolation of the member forces at the output points as described in previous section. Interpolation for the heels of the U-frame is accomplished in the same manner as for the other member since internally the member forces are always computed at 11 points, although member force output is only given at 3 points for the heels.

157. The details of the stress calculations are described in the next section. However, the output stresses and their sign convention are summarized here. First, the stresses due to axial force and bending moment are output as follows. The maximum compressive stress in the concrete (compression positive) is computed on the side of the member in which the moment induces compression. The maximum stress in the outer layer of compressive reinforcement (compression positive) is computed if compressive reinforcement is specified. The maximum tension stress in the steel (tension positive) is computed in the outer layer of tension steel specified.

158. If no tension steel is specified, the maximum tension stress in the concrete (tension positive) is computed on the side of the member in which the moment induces tension. A warning message is also printed if no tension steel is specified at the section to ensure that the user has placed the steel on the intended side. The user should thoroughly review the stress situation if it is intended to omit steel on the tension face for any loading. The concrete shear stress is always output as a positive quantity. If the direction of the shear stress is desired, the user can refer to the output of member shear forces.

159. In addition to the stress output described above for the individual members, the maximum stresses at each section investigated by the user are saved and summarized in Section 0.2 of the output. Stresses are output for evaluation of the user. No comparisons of the computed stresses are made with the allowable stresses in the investigation mode. The input allowable stresses have no effect on the program solution in the investigation mode.

Working Stress Calculations

160. Stresses due to flexure and axial forces are computed using the simple equations traditionally used for the working stress design option. For combinations of axial force and moment that do not produce tension, the gross transformed section properties are used. For cases involving tension, the cracked transformed section properties are used. Figure 39 shows a typical member rectangular cross section for either wall or base slab member. The thickness H varies. However, the width is always 12 in. Up to three layers of steel may be on both the tension and compression sides.

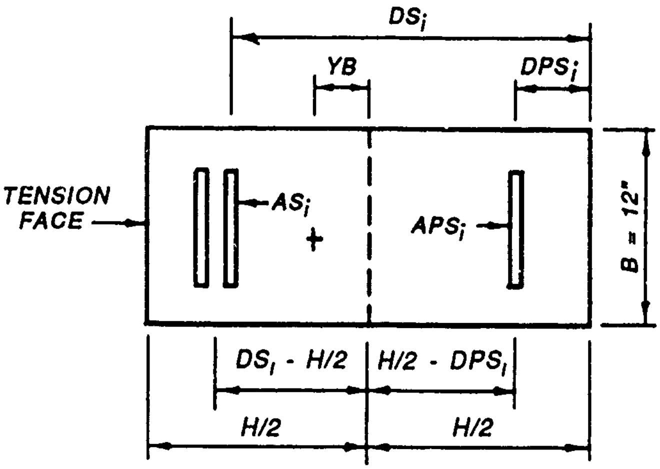
161. Tension steel is defined by area AS(I) and distances DS(I); compression steel is given by areas APS(I) and DPS(I). The modular ratio of the tension steel is RNS, and the modular ratio of the compression steel is RNPS where RNPS is taken as 2*RNS. The modulus of elasticity of the steel is taken as 29,000 ksi.

162. It is initially assumed that the entire section is in compression as shown in Figure 39b. Thus, the transformed gross section may be used for simple "P/A + Mc/I" calculation of stress. The axial force at middepth is RNP, and the moment about middepth is RMP. In the following equations, all summations are take over all steel layers. The gross transformed section area, AT, is found by

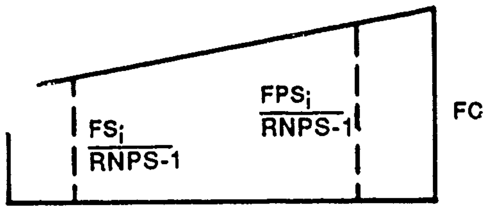
$$AT = (RNPS-1)*[\Sigma\ AS(I) + \Sigma\ APS(I)] + B*H$$

The distance from middepth to the centroid, YB, is found by

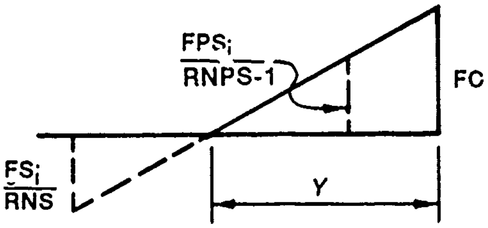
$$YB = (RNPS-1)*(\Sigma\ \{AS(I)*[DS(I)-H/2]\}-\Sigma\ \{APS(I)*[H/2-DPS(I)]\})/AT$$



a. Cross section



b. Compression controls



c. Tension controls

Figure 39. Stress variation on transformed sections

The gross transformed moment of inertia, RIT, is given by

$$RIT = B*H^3/12 + B*H*YB^2 + (RNPS-1)*$$

$$(\Sigma\ \{AS(I)*[DS(I)-H/2-YB]^2\} + \Sigma\ \{APS(I)*[H/2-DPS(I)+YB]^2\})$$

The concrete stress, FC, compression positive, is given by

$$FC = RNP/AT + RMC/RIT*(H/2 + YB)$$

where

$$RMC = RMP + RN*YB$$

The compression steel stress in the Ith layer, compression positive, is given by

$$FPS(I) = \{RNP/AT + RMC/RIT*[H/2-DPS(I)+YB]\}*RNPS$$

163. If no tension steel was specified on the tension side of the member then the maximum tension stress in the concrete is computed and output along with a warning message that no tension steel was specified. The maximum concrete tension stress, FCT, is computed by

$$FCT = -RNP/AT + RMC/RIT*(H/2-YB)$$

If tension steel is present, the tension steel stress in the Ith layer, tension positive, is computed by

$$FS(I) = \{-RNP(I)/AT + RMC/RIT*[DS(I)-H/2-YB]\}*RNPS$$

If all the steel layers are in compression, then the stresses computed as described above are assumed correct. If any of the steel layers are in tension, the solution is repeated for the tension solution which follows.

164. The tension solution is the same plan as presented by Shushkewich (1983) except that it was modified to allow eccentricities within the depth of the section. The axial force, RNP, and moment, RPM, at the centroid are

transferred to the corresponding values at the compression face, RN and RM (Shushkewich 1983).

$$RN = RNP$$
$$RM = RMP - RNP*H/2$$

165. The distance to the neutral axis Y is found by summing forces and moments and gives rise to a cubic equation similar to one by Shushkewich (1983) except that it provides for multiple layers.

$$1/6*B*RN*Y^3 + 1/2*B*RM*Y^2 + (BET*RN + ALP*RM)*Y - (GAM*RN + BET*RM) = 0$$

where

$$ALP = \Sigma\ RNS*AS(I) + \Sigma\ (RNPS-1)*APS(I)$$
$$BET = \Sigma\ RNS*AS(I)*DS(I) + \Sigma\ (RNPS-1)*APS(I)*DPS(I)$$
$$GAM = \Sigma\ RNS*AS(I)*DS(I)^2 + \Sigma\ (RNPS-1)*APS(I)*DPS(I)^2$$

The equation is solved for Y by Newton's iterative solution. Once the value of Y is found, the concrete stress, compression positive, is found from

$$FC = RN*Y/(B*Y^2/6 + ALP*Y - BET)$$

For the case of no axial force, the governing equation becomes a quadratic and Y is found by

$$Y = [(ALP^2 + 2*B*BET)^{0.5} - ALP]/B$$

Then, the concrete stress is found by

$$FC = RM*Y/(GAM - BET*Y - 1/6*B*Y^3)$$

Next, using similar triangles the steel stress is seen to be

$$FS(I) = RNS*[DS(I)-Y]/Y*FC$$

and

$$FPS(I) = RNPS*[Y - DPS(I)]/Y*FC$$

If any of the compressive layers are in tension, then a second solution is made with the tension modular ratio used for the layers which were in tension.

166. The above solution assumes some compression exists in the concrete. For larger values of axial tension, the value of Y is negative and the concrete is assumed completely ineffective. By referring to Figure 39 and omitting the concrete, the properties of the steel beam are computed as

$$YBT = [\Sigma\ APS(I)*DPS(I) + \Sigma\ AS(I)*DS(I)]/AT$$

where

YBT = distance to the centroid of the steel from the "compression" face.

AT = Σ APS(I) + Σ AS(I).

and the effective moment of inertia is expressed by

$$RINT = \Sigma\{APS(I)*[YBT-DPS(I)]^2\} + \Sigma\{AS(I)*[YBT-DS(I)]^2\}$$

The stress in any layer of the tension steel is given by

$$FS(I) = RN/AT + RMT*[DS(I) - YBT]/RINT$$

where RMT, the moment about the centroid, is expressed by

$$RMT = RM + RNP*YBT$$

167. The above solution is used when the Newton solution fails provided there are at least two layers of steel. If the Newton solution does not converge and there is only one layer of steel, then no solution is possible and the steel stress is set equal to 999.99 ksi. A solution will be found for any combination of steel layers totaling two or more, even two layers on the same face. However, the stresses would be quite high if there was any significant moment about the centroid of the two layers.

168. The nominal shear stress as a measure of diagonal tension is computed by dividing the shear force by B*DSH, where B is 12 in. and DSH is the depth to the centroid of the tension steel. However, for sections without tension steel, DSH is taken as 80 percent of the total depth of the section.

It should be noted that stresses computed are nominal at best and that shrinkage effects have been ignored. Thus, cases without tension steel specified should be thoroughly reviewed, and appropriate action taken to prevent possibly excessive tension stresses.

Review of Member Strengths in Investigation Mode - SD Option

169. Using the strength design option, section strength capacities may be reviewed at the predetermined locations described earlier. The flexural-axial capacities are calculated using the procedures outlined in ETL 1110-2-312 (Headquarters, Department of the Army 1988) or ACI 318-83 (1983). Actual calculations for section strength are made using subroutines taken from the CASE program CSTR (Hamby and Price 1984).

170. The primary input for the strength design option is as follows:

a. FPC = standard ultimate concrete strength in compression.

b. WTCONC = unit weight of the concrete in pounds per cubic foot.

c. FY = yield stress of the steel in tension and compression. (A limit may be placed on this value depending on the design criteria chosen.)

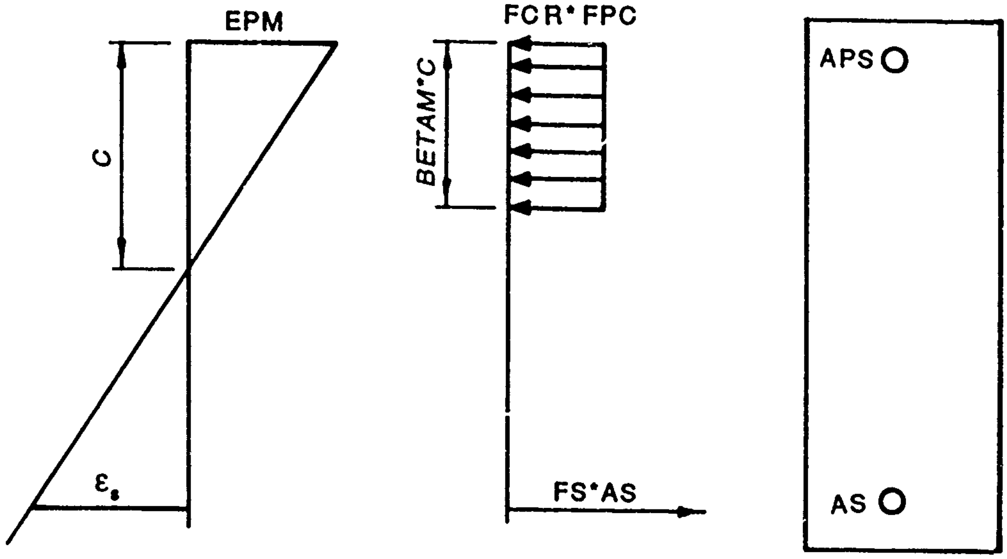
d. PBRAT = ratio of steel permitted to that associated with a balanced condition. (A limit may be placed on this value depending on the design criteria chosen.)

e. 'DCRIT' = design criteria. 'DCRIT' = "HYD" for Corps Hydraulic Concrete Structure design criteria.

f. 'DCRIT' = "ACI" for ACI Code design criteria. 'DCRIT' = "INP" to input the parameters defining the design criteria.

171. If the program user chooses the "HYD" or "ACI" options, then it is not necessary to specify the parameters that define the design criteria. The design criteria are described with reference to Figure 40. The figure shows a typical section and the associated strain and stress distribution assumed for calculations at failure conditions. Table 1 shows the values of the parameters and the maximum values of FY and PBRAT for the "HYD" and "ACI" options.

172. As illustrated in Figure 40 and used in Table 1, the maximum strain in the concrete is EPM. BETAM is the ratio of the depth of the rectangular stress block to the depth of the neutral axis. FCR is the ratio of the assumed uniform stress in the calculation stress block to FPC. In addition, the following parameters are used. PMAXF is the ratio of the maximum useable



a. Strain   b. Stress block   c. Section

Figure 40. Strength design conditions

Table 1

Parameters Defining Strength Design Criteria

Parameter	FY(max)	PBRAT(max)	EPM	BETAM	FCR	PMAXF	PHIA	PHIF	PHIS
"HYD" value	48.0	0.25	0.0015	*	0.85	0.7	0.7	0.9	0.85
"ACI" value	--	0.75	0.0030	**	0.85	8.0	0.7	0.9	0.85

* Depends on FPC and varies between 0.55 and 0.5.
** Depends on FPC and varies between 0.85 and 0.65.

compressive force to the force on the interaction curve for no eccentricity. This factor is used to account for unavoidable eccentricities which may occur even when the calculations indicate no appreciable eccentricity. PHIA is the capacity reduction factor phi for the pure axial load state. PHIF is the capacity reduction factor for the pure flexure case. PHIS is the capacity reduction factor for shear.

173. It is anticipated that the user of the program will normally use the "HYD" or "ACI" criteria depending on whether or not crack control is essential. It should be noted that if the "ACI" option is chosen, the ACI crack control criteria are not considered. The "INP" option is included

primarily for possible parameter studies on the effects of the design criteria on the results.

174. A primary output of the program is the ratio of the flexural-axial capacity required based on the factored axial force and bending moment at the section to the flexural-axial capacity provided by the section. A value of 1.0 indicates that 100 percent of the section's capacity is utilized. The appropriate phi factors are considered. Thus, a value of one or less indicates the strength of the section is satisfactory.

175. Load factors are input separately for each EM-like load case and any special load case that may exist. A single load factor is input for each load case and is applied to the results of the analysis for all loads. Thus, no distinction is made between dead and live loads. This approximation is slightly conservative. However, the loading which governs the design of U-frame structures is so predominantly live, in nature, that this approximation will have very little if any affect on the final results. It is anticipated that the user of the program will specify the normal live load factor as the load factor.

176. It should be noted that the basic frame analysis is made for the nominal or unfactored load level. Where nonlinear response is important, such as for investigations made specifying the nonlinear force-deformation solution for wall pressures, this use of unfactored load levels may not be appropriate. For such nonlinear investigations, it is recommended that the user of the program "apply the load factor" directly to the loading conditions with lower strengths of the soil properties or conversely higher pressures on the active side of the loading curves. Likewise, water tables should be raised to represent the limit state condition, and then a load factor of one used. Thus, the nonlinear analysis would be made at the limit state condition. It should be emphasized that this procedure is only for specific nonlinear investigations and that for all designs and routine investigations the standard procedure for describing the nominal loading and a live load factor on the order of 1.9 for a "HYD" design or 1.7 for a "ACI" design should be used.

177. The evaluation of the adequacy for combinations of axial load and flexure is made on the basis of the flexural-axial strength interaction curve. Figure 41 shows a typical curve that might be computed for a "column" section with symmetrical compression and tension reinforcement. It is assumed that the phi factors have already been applied. Likewise, the limiting axial force

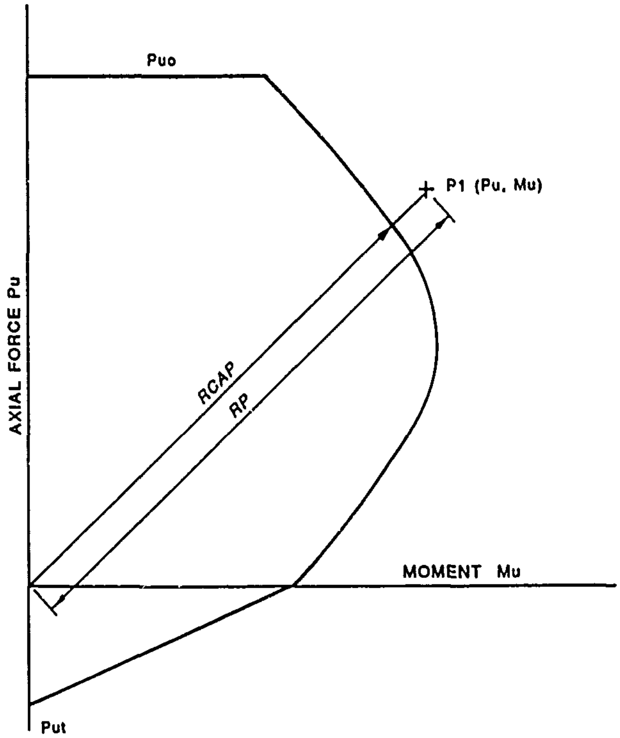


Figure 41. Interactive curve for "column" section

in compression has been calculated creating the horizontal cap on the curve. Thus, the curve represents the maximum useable strength, and combinations of axial load and moment at a section that create a point inside the curve represent a safe or acceptable solution.

178. Let P1 represent a point describing the factored load conditions. RP is the distance to the point from the origin. RCAP is the distance to the point on the curve having the same eccentricity as the actual loading. Thus, we define the strength ratio, STRAT, as the ratio of the required strength to the strength capacity or

$$STRAT = RP/RCAP$$

Clearly a value of STRAT less than or equal to one implies that the strength is adequate. Note that this essentially radial ratio, STRAT, is sufficient to define the strength adequacy for all cases for sections with shapes as shown in Figure 41 (i.e., it works even for cases of small or zero moment in either the axial compression or axial tension zones).

179. For sections with significantly more tension steel than compression steel, the normal case for design of U-frame structures, the shape of the interaction curve will be as shown in Figure 42. The strength ratio must still be less than or equal 1.0. In addition, due to the unsymmetrical nature of the reinforcement, the loading must be such that for axial loadings in tension, the slope of the line from the origin to Pu,Mu is at least as large as the slope of the line connecting the origin and the point on the interaction curve with the maximum axial tension capacity, Put. The program checks for this required slope, and if it is not met, the value of STRAT is set equal to 99.99 that is well in excess of the maximum limit of 1.0.

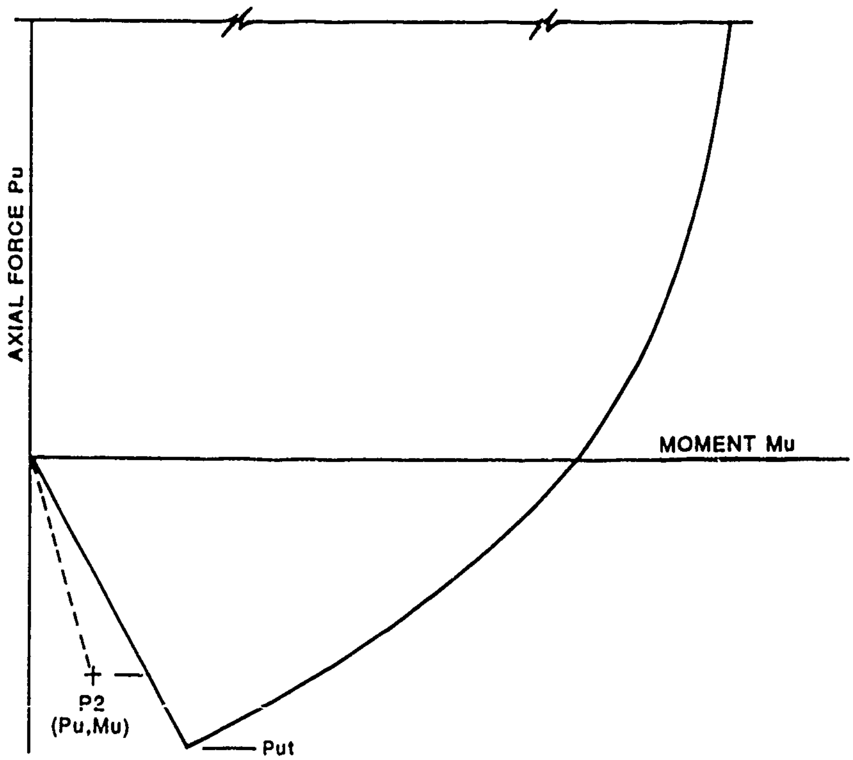


Figure 42. Interaction curve for "beam" section

180. In addition to the strength ratio at the section, a ductility ratio is also output. The ductility ratio is computed to give an indication of whether or not the section has sufficient size such that the amount of tension steel is less than the amount for a balanced failure and should be less than or equal to one. The value of ductility ratio computed in the investigation mode is the ratio of the moment acting on the section to the moment capacity of a section with PBRAT times the area of tension steel

corresponding to the balanced conditions. The balanced conditions are defined by having the strain in the tension steel equal to its yield value simultaneously with the attainment of a compression strain of ECB = 0.003. Note that ECB is similar to the value of EPM at actual failure conditions. However, in computing the ductility ratios the value of ECB is used for the maximum concrete strain regardless of EPM. Likewise, in computing the ratio of the depth of the stress block to the depth of the strain block, for the balanced condition, the value of ß1 computed with criteria from ACI (1983) is used regardless of the value of BETAM.

181. If, for any load condition, no steel is specified on the tension side of the member, a warning message will be indicated. It is possible that for small values of moment, the strength and ductility requirements may be satisfied. However, the user of the program is cautioned that such a condition could imply very large strains, and hence excessive cracking is possible.

182. The nominal shear capacity VCN of the section is computed for members with compressive forces Pu by

$$VCN = 2[1. + Pu/(2000*AG)]*12*DSH*FPC^{0.5}$$

and for member in tension by

$$VCN = 2[1. + Pu/(500*AG)]*12*DSH*FPC^{0.5}$$

where AG is the gross concrete section, and DSH is generally the flexural depth at the section. However, if the program user does not specify any steel on the tension face, DSH is taken as 80 percent of the total depth. The user is warned that the application of the above equations to cases with no tensile steel is not guaranteed to produce adequate results since shrinkage and other tension producing factors are not considered. Pu is take as positive in compression.

Omission of Symmetrical Output

183. Detailed pressure and member force output are listed only for the members on the left side of symmetrical U-frames under symmetrical EM-like loadings. However, if the loading involves special load cases or the load-

deformation option for wall pressures, detailed output will be given for all members. Likewise, investigation results of stress or strength criteria are available for the right-side members of symmetrical U-frames only for unsymmetrical EM-like load cases, special load cases, or when using the load-deformation option for wall pressures.

PART VI: DESIGN MODE

General Description

184. It is possible to design by a trial and correction process using the investigation mode. However, this method is often tedious and time-consuming. Thus, it is desirable to have a design mode for the program. The design module was developed with the guidance of engineers experienced with the design of basins and channels and could be considered to be something akin to an "expert system." However, it should be noted that any automated design procedure will have a large number of design decisions programmed. Such decisions, while generally providing a safe and reasonable structure, will not always guarantee the most economical structure. In addition, designers must be certain that any limitation of the program, which may be insignificant for most U-frame structures, will not affect the validity of the design of their particular structure. Thus, it is essential that the user of the program understand the design algorithm included in the program. In addition, it is necessary that the user of the program in the design mode be familiar with the investigation features of the program previously described. The design mode is simply a specified procedure of executing a series of analyses and checks to arrive at a final solution.

185. The program requires that the designer specify a minimum cross section of the U-frame. This decision by the user can obviously have a considerable effect not only on the final design but also on the computer cost of the computer-aided design. If the designer specifies a larger section than needed, then the program will simply select reinforcing for that size structure. On the other hand, if the user specifies too small an original section, then a design solution may not be reached. The program does not allow an unlimited amount of incrementing sizes, which could cause excessive computer costs. However, if the design criteria cannot be satisfied, within the iteration limits permitted, the program will allow the user to obtain output which will give pressures, forces, and stresses, or a review of the strength criteria for the last design attempted. This procedure will allow the user to make a better selection for the next design run. The limits which are placed on the design iterations are described subsequently.

Design Mode Restrictions

186. The program is structured such that the data input and procedures are as close as possible for the design and investigation modes. However, several restrictions were placed on the design mode to avoid unnecessary complications of the design algorithm for cases rarely encountered. These restrictions also tend to simplify the input for the design mode. The restrictions on the design mode are listed below.

187. First, the geometry of both basin and channel structures will be completely symmetrical for the design mode. Next, input dimensions are either fixed or else the minimum for design iterations.

188. Then active loadings, with only one exception (see paragraph 189b), must be symmetrical EM-like load cases in the design mode. Loads permitted include some but not all of the loads allowed in the investigation mode, described in Part IV. Loads allowed in the design mode are discussed below.

Active Loading for Design Mode

189. The types of active loading allowed include:

- a. Self-weight of concrete U-frame automatically generated from geometry of section (updated during design) and input unit weight.
- b. Hydraulic loading wherein all hydraulic pressure. are automatically computed from the input water elevatic drain locations, and specified efficiencies of the drai elevations must be symmetric; except for two-ba, basins and channels, the internal water elevations may be unsymmetrical. This exception was made to allow for the design of the internal wall. However, it should be noted that the program still only designs the left "half" of the structure. The designer is responsible for ensuring that sufficient load conditions are specified if the unequal internal water elevations control the design of any member other than the central wall.
- c. Active earth pressure by wedge solution.
- d. At-rest pressures by modification of active wedge pressures by input coefficient.
- e. Vertical surcharge loads as part of wedge solution.
- f. Empirical wall and heel pressures computed from input soil elevations and lateral pressure coefficient.

Reactive Loading for Design Mode

190. The types of reactive loading allowed include:

   a. Base slab pressures computed using compression only beam on elastic foundation model, i.e. distributed vertical elastic springs acting only in compression.

   b. Vertical anchor forces computed as tension only elastic spring model. (See subsequent discussion of uplift.)

   c. Beam slab pressures computed by statics with user specified shape. This procedure is similar to a "P/A" + "Mc/I" approach except the shape of the "P/A" portion can be specified.

   d. Base shears computed to satisfy horizontal equilibrium from all active forces uniformly distributed either over the base or on the basis of distributed horizontal springs on the base slab.

Reinforcement by WSD or SD Options

191. The sections are sized and reinforcement is selected based on shear, flexure, and axial force effects as described herein, and no consideration is given to bond, anchorage, or detailing requirements. The ACI strength design criteria for cutting off steel in a tension zone, the minimum amount of tension steel needed to avoid a possible flexural cracking failure, and distribution of steel to avoid oversize cracks are not checked. Also, it is assumed that the depth-span ratios are such that consideration of the deep beam theory is not required. Channel heels are normally very short in length and should be designed with due consideration of their high depth-to-span ratio. Thus, no consideration is given to the design of channel heels by the program.

192. In the investigation mode, the stresses are computed, or strengths are evaluated at user specified points. However, the design mode computes the required areas of steel at certain predetermined points (usually the tenth points of members). Consequently, user input is reduced considerably in the design mode. Figures 43 and 44 illustrate the reinforcement input for the design mode of basins and channels, respectively. As shown in Figure 43 for basins, clear cover is generally specified in four locations, (COVER (I),I = 1,4). The center-to-center spacing between parallel layers of steel, CCLAY, is constant. The maximum number of layers of tension reinforcement are

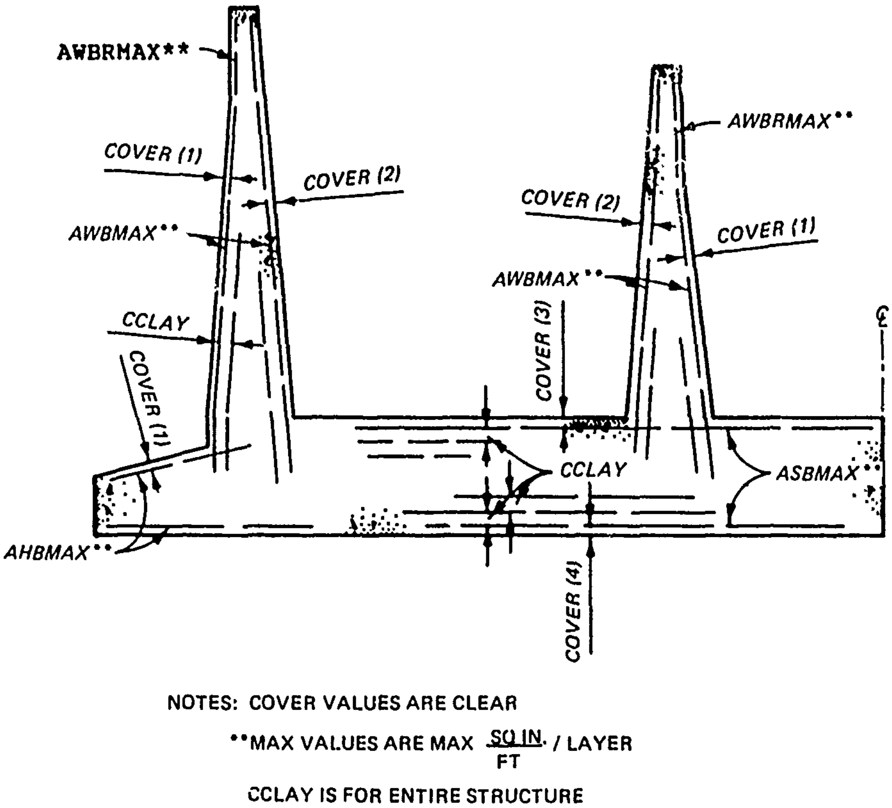


Figure 43. Reinforcement description for design mode, basins

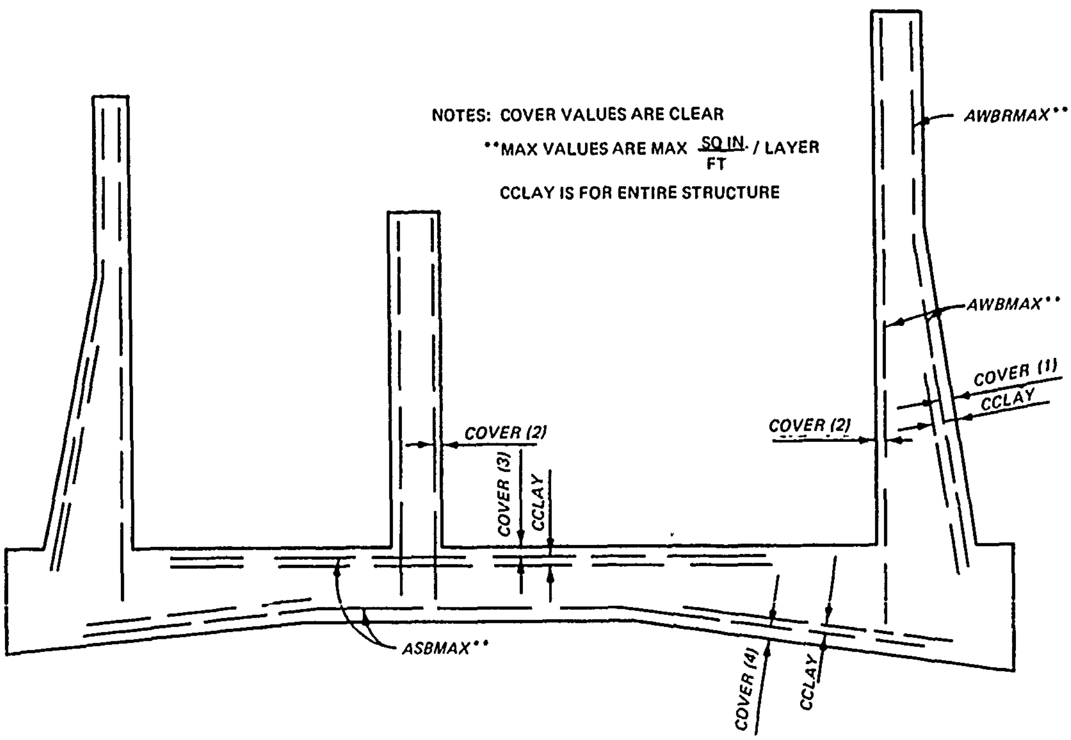


Figure 44. Reinforcement description for design mode, channels

specified for the walls, slab, and heel, NOLAYW, NOLAYSB, and NOLAYH, respectively. The maximum number of layers above the break in the wall is limited to one. Then the maximum amount of steel per layer is specified by giving the area in square inches per foot, using the variables AWBRMAX, AWBMAX, ASBMAX, and AHBMAX for the walls above the break, below the break, base slab, and heels, respectively, as shown in Figure 43. The maximum diameter must also be given in these same locations by specifying DWBRMAX, DWBMAX, DSBMAX, and DHBMAX. If the heel is absent, then the data normally required for the heels are omitted. Details on required input are included in Volume B.

193. Figure 44 for channels is almost identical to the reinforcement details in the previous figure for basins except that the channel figure does not show any reinforcement for the heels which are not designed by the program. Complete input details are given in Volume C.

194. The steel is assumed to fill up the outer layers first in computing the effective depth of the member. Figure 45 illustrates this procedure. The figure shows partial input and output for a channel. As seen in input Section I.5, the base slab can have a maximum of two layers (NOLAYSB = 2) with a maximum area of steel of 2.00 sq in./ft in each layer (ASBMAX = 2.00). Output section O.2 shows that member number 2, which is the base slab, requires two layers near the left end (DISTANCE = 0,2.4) and near the center of the symmetrical member (DISTANCE = 9.6,12.0).

195. The selected output for member 11 (wall) shows that no steel is required based on stress or strength calculations at the top of the wall, and two layers are required at the base. Again, it should be emphasized that the steel areas shown are those based on stress or strength calculations for flexure and axial force at the indicated section. The steel has to be extended past the points shown to ensure proper anchorage, and good detailing practice should be followed.

196. The user is also reminded that the program does not specify a minimum area of steel based on temperature, shrinkage, or prevention of a cracking failure (ACI 318, paragraph 10.5.1). However, the program will output a nominal value of 0.01 sq in. on the side, or sides, of a section for which an applied moment tends to cause tension, even if the stress or strength calculations show that no steel is required on that face.

197. Figures 46 and 47 show graphical output of the required areas of

```
   I.5 REINFORCEMENT FOR DESIGN OPTION

 NUMBER OF LAYERS
 WALL     SLAB
 NOLAYW  NOLAYSB
    2        2

CLEAR COVER AND CL TO CL LAYER DISTANCE(CCLAY)
COVER (IN)                          CCLAY(IN)
COVER(1) COVER(2) COVER(3) COVER(4)    CCLAY
    2.50     3.00     3.00     3.50     4.00

MAXIMUM AREAS PER LAYER AND DIAMETERS
 WALL BELOW BREAK          SLAB
    AREA    DIAM.     AREA    DIAM.
  AWBMAX   DWBMAX   ASBMAX   DSBMAX
 (SI/FT)     (IN)  (SI/FT)     (IN)
    2.37     1.00     2.00     1.13

-------------------------------------------
O.2 SUMMARY OF STEEL REQUIREMENTS BY MEMBER

            ************* MEMBER  2 *************

             ********** TOP STEEL **********
                        NONE REQUIRED FOR STRENGTH

            ********** BOTTOM STEEL **********
DISTANCE    BAR         AREAS (SI/FT)    STEEL RATIO  DEPTH(D)
    (FT)  DIAM.                BY LAYER     AS/12*D       (IN)
           (IN)        1       2      3
    0.00  1.128     2.00     .51                .0076      27.57
    2.40  1.128     2.00     .15                .0070      25.45
    4.80  1.128     1.95                        .0070      23.32
    7.20  1.128     1.94                        .0076      21.19
    9.60  1.128     2.00     .32                .0101      19.06
   12.00  1.128     2.00    1.75                .0184      16.94

            ************* MEMBER 11 *************

            ********** TOP STEEL **********
DISTANCE    BAR         AREAS (SI/FT)    STEEL RATIO  DEPTH(D)
    (FT)  DIAM.                BY LAYER     AS/12*D       (IN)
           (IN)        1       2      3
   20.00                                       0.0000       7.00
   18.00                                       0.0000       8.60
    -    -    -    -    -    -    -    -    -    -    -    -
    4.00  1.000     1.50                        .0063      19.80
    2.00  1.000     2.30                        .0090      21.40
    0.00  1.000     2.37    2.15                .0164      23.00
```

Figure 45. Sample design mode reinforcement input/output

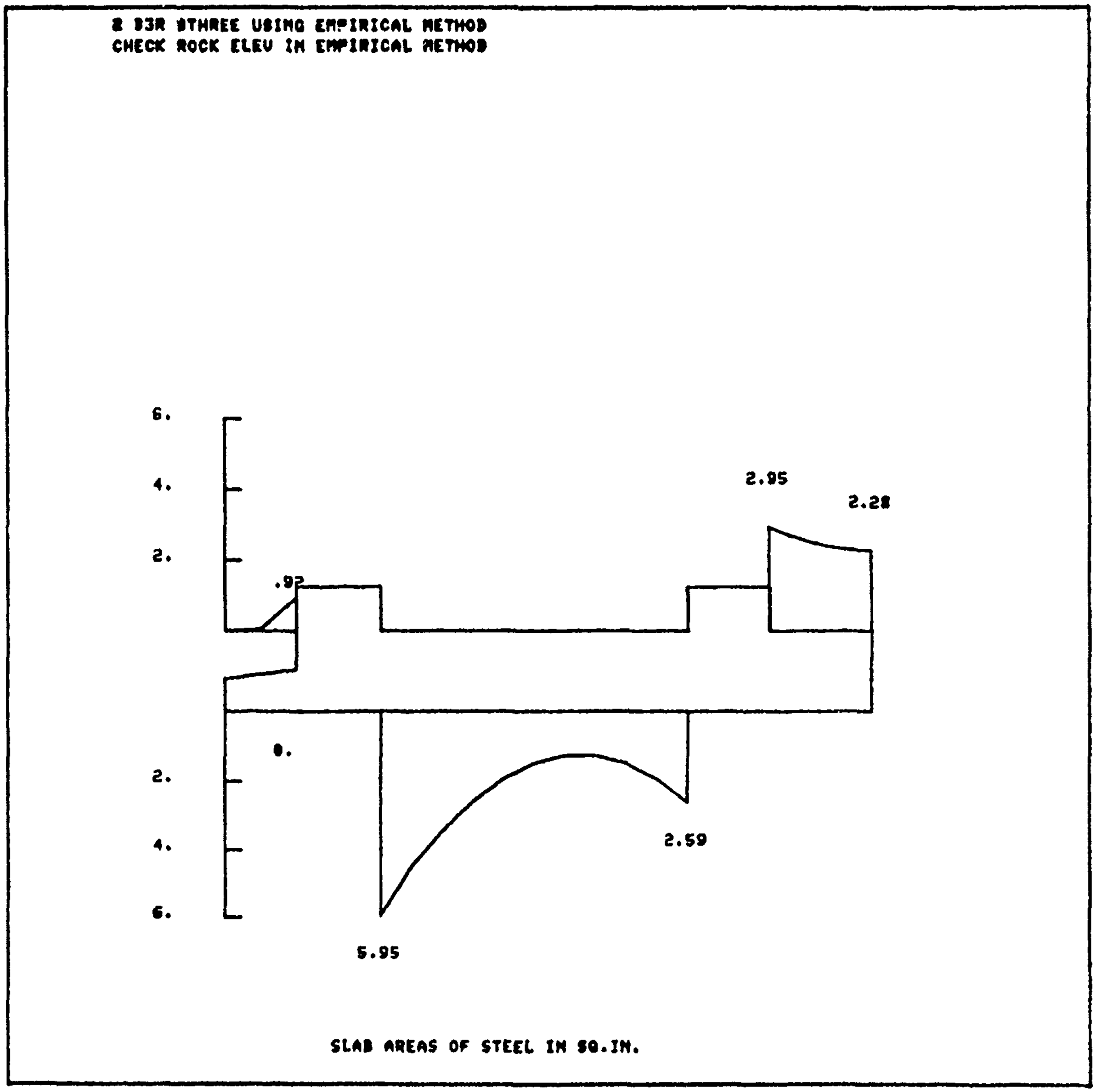


Figure 46. Sample area of steel plot for base slab

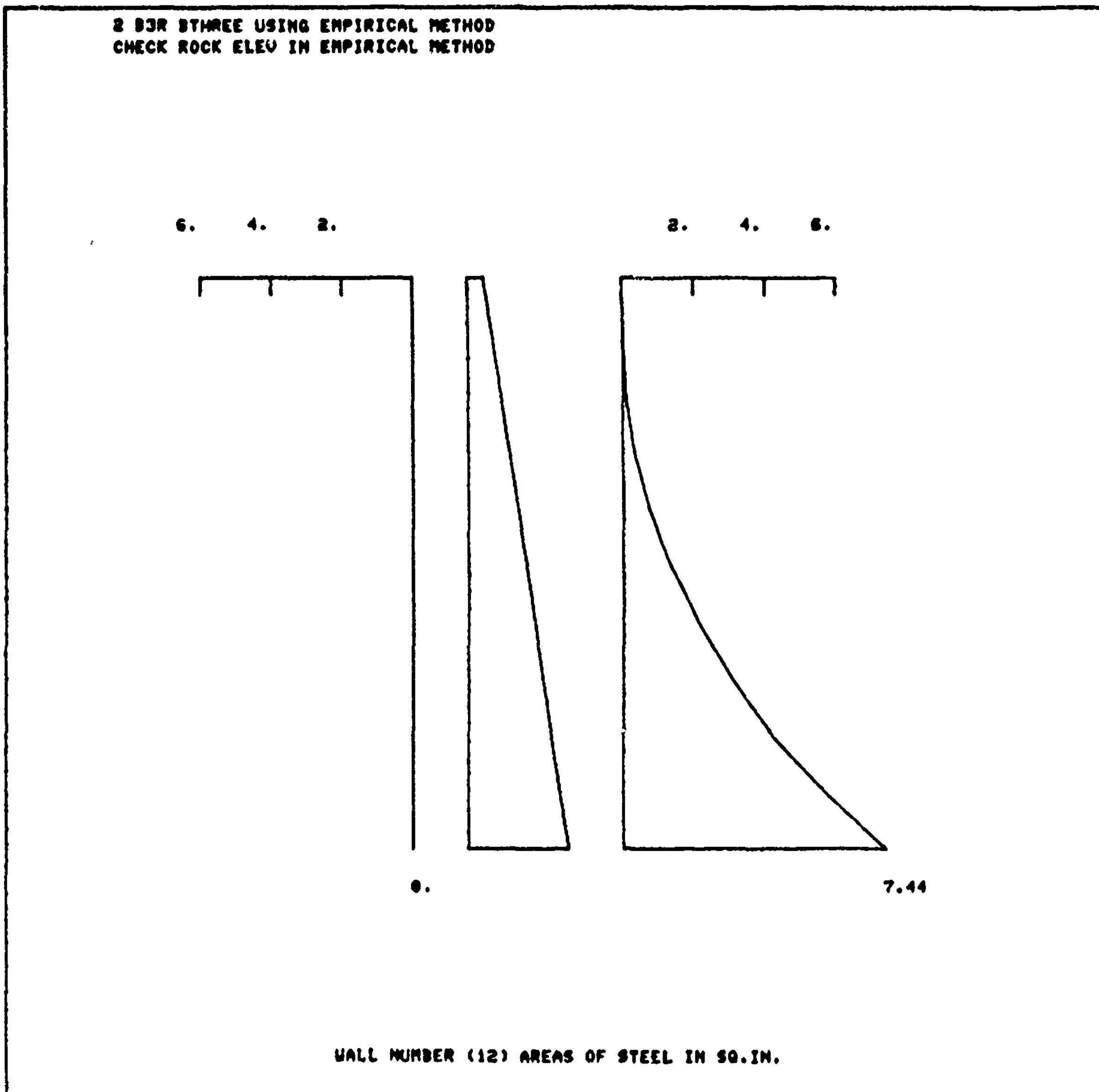


Figure 47. Sample area of steel plot for wall

steel for a base slab and a wall member, respectively. The required areas are plotted on the sides of the member for which steel is needed based on axial-flexural requirements. While not shown in the example output, U-frames subjected to several loading cases or significant axial tension forces may often require steel on both sides of a member.

Design Criteria - WSD Option

198. When designing by the WSD option, basic allowable stresses are input, and then an allowable stress multiplier is input for each EM-like load case. For instance, to allow a 100 percent stress increase in the allowable stresses for a certain EM-like load case an allowable stress multiplier of 2.0 would be input for that EM-like load case.

199. Design for flexure and axial force is based on actual computed stresses being less than allowable stresses at critical sections described subsequently. Actual stresses are computed using allowable stress equations described in the earlier investigation discussion. Stresses computed and their corresponding allowables are concrete compression (FC and FCA), steel (FS and FSA), and shear (VC and VCA).

200. For economy, it is generally desirable that the total amount of steel be less than that corresponding to balanced conditions. To ensure this condition is satisfied, the minimum depth required for balanced conditions, DBAL, (including effect of axial force) is computed as described subsequently, and the actual value of D is kept at least as large as DBAL.

201. In addition to checking that FC does not exceed FCA and FS is less than or equal to FSA, the program checks short column capacity by requiring the axial force, P, not to exceed the axial force corresponding to a stress of FCA on the extreme compression side and 0. on the tension side. This condition defines PO where

$$PO = .5*FCA*AG$$

where AG is the gross concrete area. If FCA is equal to .45*FPC the result is

$$PO = .225*FPC*AG$$

which is almost identical to the limiting axial force specified by ACI 318-56 (1956). Long-column effects are ignored.

202. Design for shear is by allowable stress provisions of ACI 318-83 (1983) for reinforced concrete members of normal depth-span ratios. Consequently, no design is done for heels of channels. The allowable shear stress, VCA, is computed by the following equations where P is the axial force:

If P is in compression (> 0.), then

$$VCA = 1.1*(1. + .0006*P/AG)*(FPC)^{0.5}$$

If P is in tension (< 0.), then

$$VCA = 1.1*(1. + .004*P/AG)*(FPC)^{0.5}$$

203. The nominal shear stress is computed as in the investigation mode, except that if the design shows no steel is needed for axial-flexural effects, DSH is computed based on one layer of steel. Thus, some minimum steel should be provided in any region of significant shear.

204. The ratios FC/FCA, FS/FSA, VC/VCA, P/PO, and DBAL/D should be less than one at all points to satisfy allowable stress criteria. The program makes these checks at the critical points, subsequently described. Also, when the user exercises the option to output the design variables during the iteration process, the values of these ratios will be displayed. This option allows the designer to be much more involved with the design process than simply taking the final results as a "black-box" solution.

Design Criteria - SD Option

205. When design for concrete and steel is by the SD option, the load factor is input for each EM-like load case as described earlier. Axial forces, moments, and shears computed at sections are multiplied times the specified load factor to check the adequacy of the sections. Design for flexure and axial force is based on the strength and ductility ratios being less than one at critical sections described subsequently. Strength and

ductility ratios are computed as described earlier for investigation of section strength.

206. For cases which calculations for axial-flexural effects show no steel is required, the effective depth for shear strength calculations, DSH, is computed assuming a single layer of steel. Thus, minimum steel should be provided at all locations of significant shear. No considerations are given to long-column effects since the axial forces in U-frames are generally quite small and the soil offers restraint against long-column effects.

207. The detailed output for the SD option shows the critical strength and ductility ratios at the output locations for all load cases. Also, the user may elect to obtain interactive output of these ratios, at critical locations, during the iterations to determine the required size of the members.

General Design Procedure

208. Permitted factors of safety for uplift and bearing are input only once per run and are constant throughout design for all EM-like load cases. Foundation size is increased to try and satisfy minimum uplift requirements. However, if the specified minimum bearing factor of safety is not achieved, no resizing of members is attempted. A warning message is displayed, and the user has the option of continuing or stopping. In general, if the criteria cannot be satisfied, the user has the option of continuing the program in order to obtain output or an immediate termination.

209. The designer should probably be generous, but reasonable, in the number of layers permitted. If the number of layers are kept low, then the total amount of steel permitted may be too low resulting in a larger concrete section than really necessary. The designer should remember that the program will automatically limit the amount of steel to the value corresponding to DBAL using the WSD option, or it will ensure the ductility requirement is satisfied when using the SD option, regardless of the maximum amount input by the designer. The user of the program may wish to experiment with varying the amount of steel permitted to do some economic parameter studies.

210. The modified half-interval procedure was developed to avoid the many wasted iterations that would occur using a simple incrementing procedure when the initial guess was well below the correct solution and still not overly penalize the experienced designer whose initial estimate is very close

to the final solution. The modified half interval iterative procedure is shown in Figure 48 for an example in which the theoretical design variable is somewhere between 20 and 21 in. The designer guessed an initial value of 14 in., and an upper limit was set at 28 in. The theoretical value of 20.5 in. could be determined by stress or strength criteria or any of the other criteria the program checks. Generally, the program sets an upper limit for a design variable at twice the initial value. Exceptions are related to the design of heels and will be discussed subsequently.

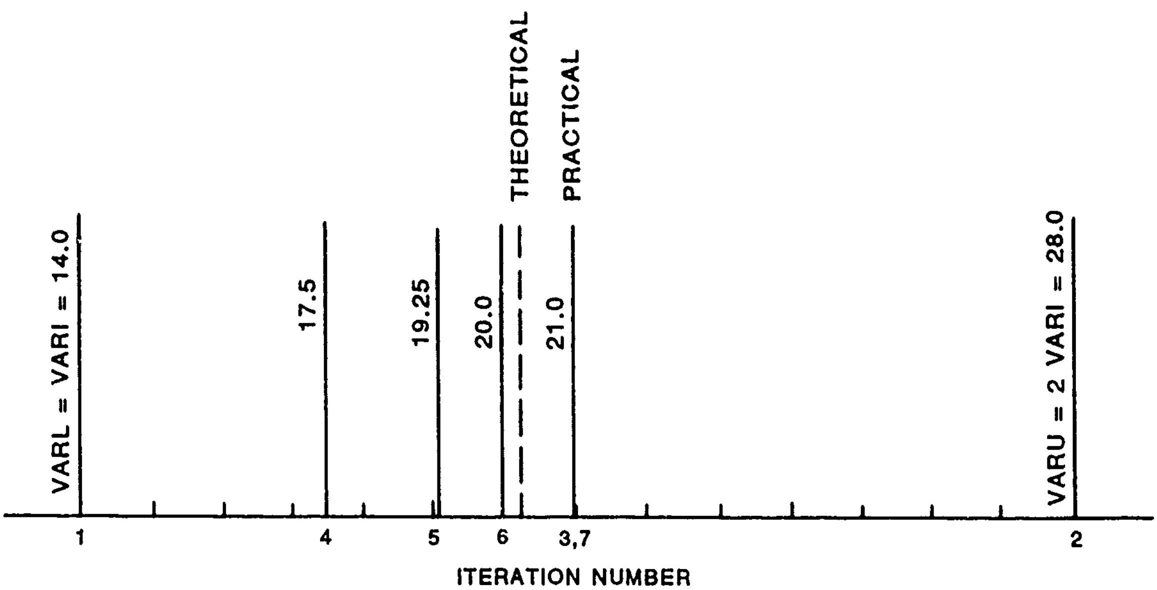


Figure 48. Modified half-interval iterative solution

211. The practical 21-in. solution would be arrived at after seven iterations as follows. First, the initial solution of 14 in. would be checked and found inadequate. Next, the upper limit of 28 in. would be checked and found adequate. If the upper limit failed, the solution could be terminated or the user could have the solution continue to obtain output, with the design variable kept at the upper limit.

212. Next, the interval between the upper and lower limit would be halved three times to reduce the interval to one-eighth its original value. At that time, the lower limit would be rounded up to from 19.25 to 20 in. and that value checked. If that solution is inadequate, the design variable would be incremented by 1-in. increments until the final solution was reached. Coincidentally, the 21-in. solution would have taken almost the same number of

iterations by the simple iteration technique (8). If the solution were closer to the lower limit, then the simple iteration procedure would be shorter than the modified half-interval procedure. However, if the upper limit of 28 in. was not satisfactory, 15 iterations would be expended by the simple iteration procedure compared to 2 for the modified half-interval scheme.

213. A brief flowchart for the design module is shown in Figure 49. The first step in the design module is to generate the structure geometry and loads using the present value of the design variables. The present values are the initial input values at the start of the program. However, these initial values are updated as appropriate during the solution. The loads generated have been described earlier and include those due to hydraulic pressure and soil pressure.

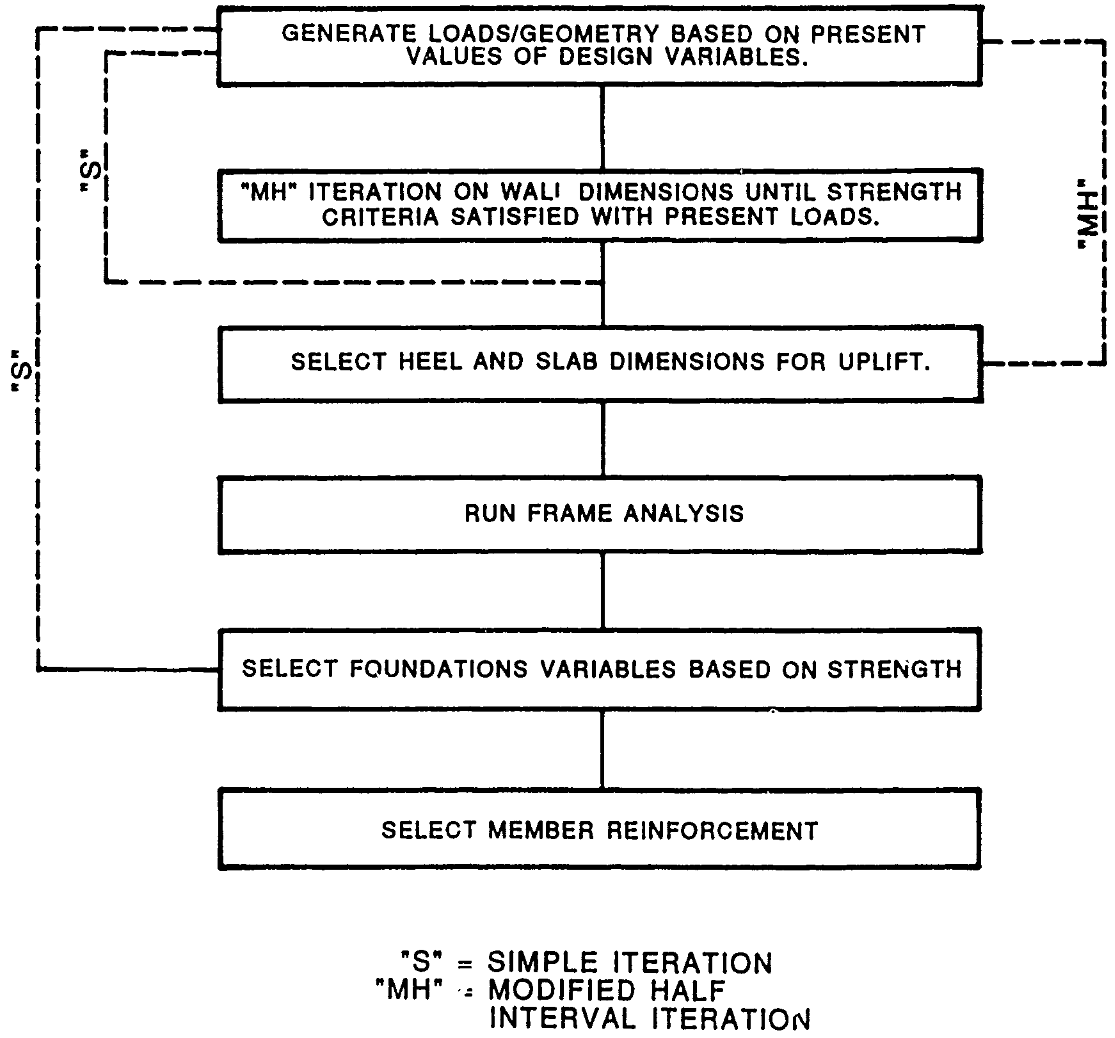


Figure 49. Design module flowchart

Selection of Wall Thicknesses

214. Next, the wall members are sized based on stress or strength criteria. The loading is assumed to remain constant and a modified half-interval iteration solution is made on the wall variables shown in Figure 50. WALLT is the thickness of the top wall and WALLB is the bottom thickness. These design variables for the wall are restricted to less than two times their initial input values. Stress or strength criteria for axial force and moment are checked at sections A-A and C-C. Shear is checked at A-A and B-B where B-B is at a distance equal to the effective flexural depth, DSH, up from the base of the foundation unless the slab provides tension support for the wall. For the rare case where the slab is in tension, the critical section for shear is at the top of the base slab. The thickness WALLT is incremented to satisfy the appropriate criteria at A-A, and then the thickness WALLB is incremented to satisfy the appropriate conditions at C-C and/or B-B.

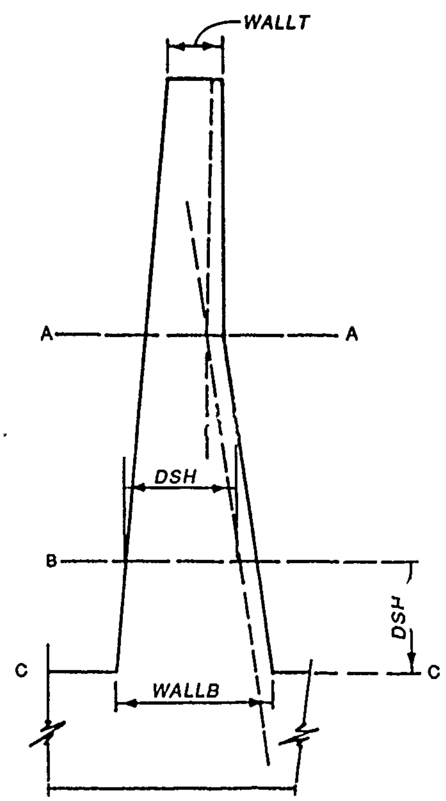


Figure 50. Incrementing wall size for strength

215. After the walls have been sized, the solution returns to the origin of the design module and recomputes the wall geometry and loads. Then the wall dimensions are checked with the new loads. Since the loads usually change only slightly as the wall dimensions increase, a simple iteration is used here (the wall dimensions are simply incremented by an appropriate increment. if necessary). An increment of 0.25 ft is generally used for basins, while an increment of 1 in. is used for channels.

Design for Uplift

216. Next, the slab dimensions are increased as shown in Figure 51 or 53 to provide the minimum desired factor of safety for uplift. If the heel dimensions are being increased, then the program returns to the start of the design module to recompute soil, water, and self-weight loads following the modified half-interval procedure as indicated in the flowchart of Figure 49. However, if only the slab thickness is being incremented, then the changes in hydraulic pressure and self-weight are computed locally during the iterations using a simple iteration procedure.

217. The incrementing procedure for uplift of basins is one of the most subjective procedures in the program. It is essential that the user understand the procedure used by the program. Different values of initial and limit values of the design variables can result in quite different designs when uplift is a major factor. Figure 51 shows how the design variables are incremented for uplift of basins. The values shown ending in I, WHEELI, DEPTHSI, DHEEL1I, and DHEEL2I, are the initial input values of the variables WHEEL, DEPTHS, DHEEL1, and DHEEL2. The user actually inputs the variables without the I suffix as indicated in the user's guide (Volume B), and the program creates the extra variables. The user also inputs a maximum heel length, WHEELM. During the incrementing procedures that follow, the slope of the heel is maintained at the value corresponding to the initial input variables.

218. When the uplift design procedure starts, the slab depth, DEPTHS, may have already been increased above the initial value, DEPTHSI, as shown with an "a" in the figure due to an increase in the corresponding wall dimension. The procedure for the initial "a" increase in the thickness of the base slab is as follows. If the wall thickness in the outer wall is increased during the design cycles based on stress or strength considerations, the base

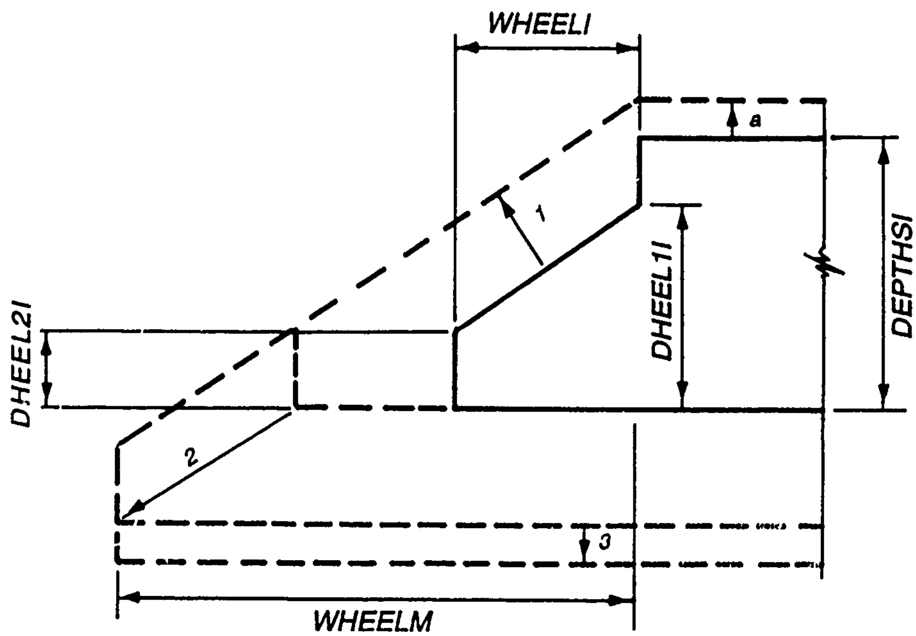


Figure 51. Uplift iterative scheme for basins

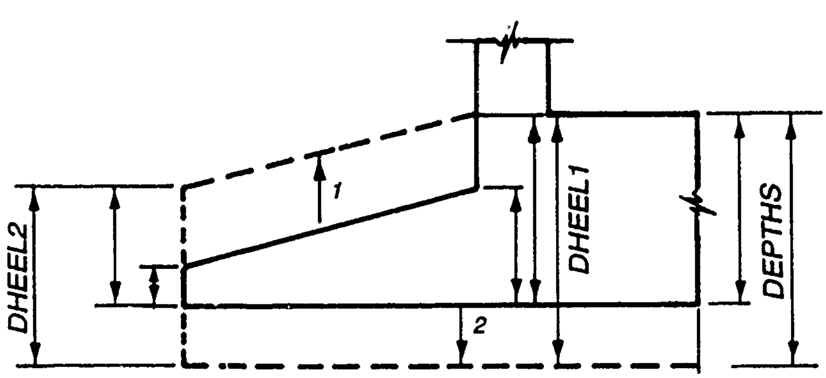


Figure 52. Stress iterative scheme for basins

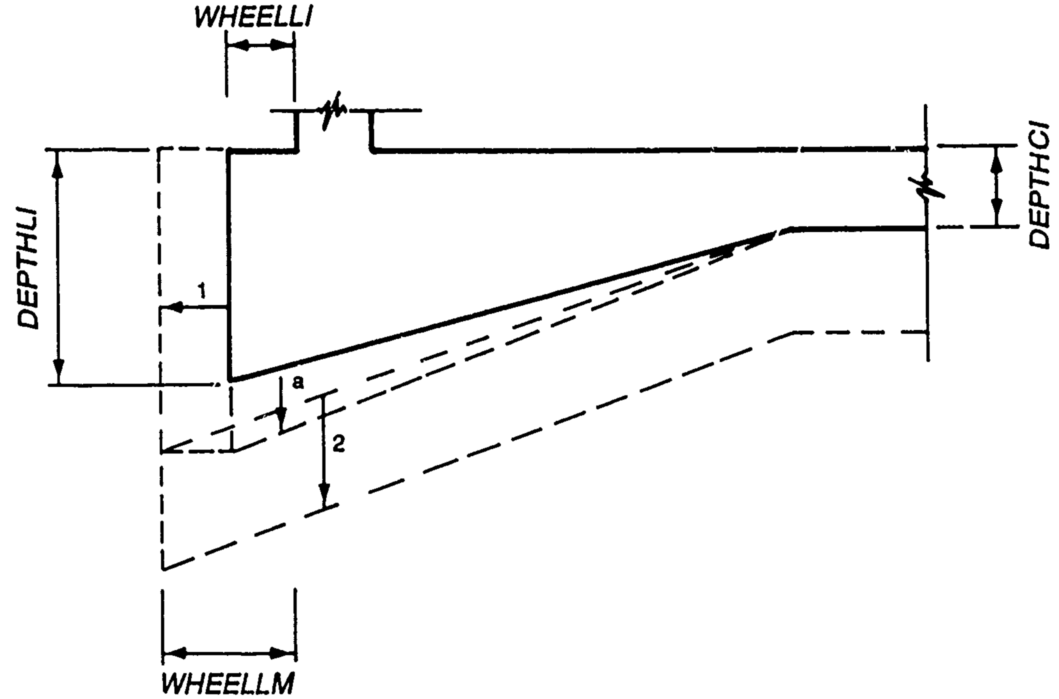


Figure 53. Uplift iterative scheme for channels

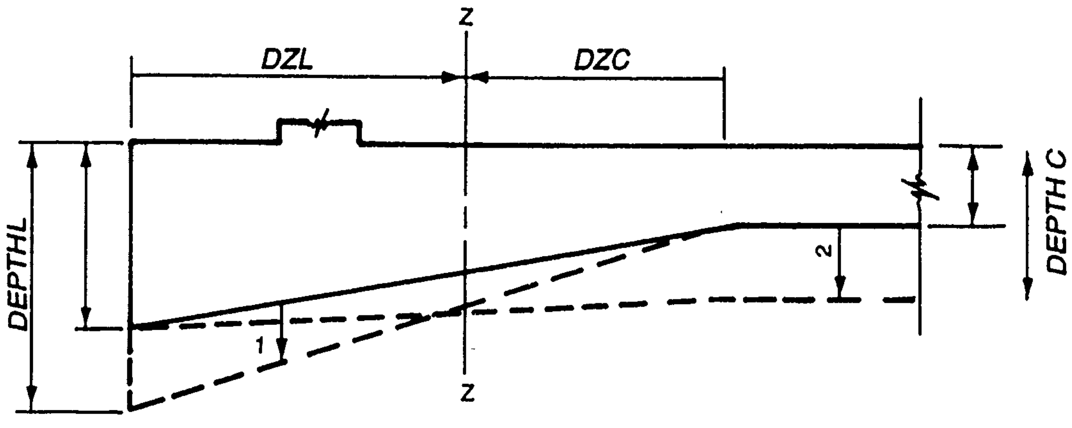


Figure 54. Stress iterative scheme for channels

slab will generally be increased by a thickness of about 75 percent of the increase in wall thickness. However, the increase will be limited such that the initial estimate of the base slab thickness does not exceed twice the value of the minimum input value. Also, the initial guess for the base slab thickness will not be increased if the wall thickness is less than the input minimum thickness of the base slab.

219. Next, the heel is increased in size until the uplift criteria are satisfied, or one of the limits shown as "1" or "2" in Figure 51 is reached, following the half-interval method. The limit on incrementing the heel, shown as "1" in the figure, is made such that the value of DHEEL1 does not exceed the value of DEPTHS. The limit on the heel incrementing procedure, indicated as "2," is made such that the value of WHEEL does not exceed the input value of WHEELM. Of course, it may be that the second limit indicated is more critical than the first.

220. If the uplift criteria cannot be satisfied with WHEEL at the above described critical limit, then the entire base slab thickness is increased using the simple incrementing procedure as indicated by the "3" until uplift is satisfactory or the value of DEPTHS reaches the limit of twice DEPTHSI. The program allows the user to obtain an output of design variables during the iteration process. It is advisable to exercise that option for basins where uplift may control in order to get a better visualization of the iteration process.

221. The incrementing procedure for uplift design for channels is shown in Figure 53. As for the channels, the input variables which vary have an I suffix, DEPTHCI, DEPTHLI, and WHEELLI, indicating these are the initial values of the corresponding variable names without the I suffix. During the wall iteration described earlier, the slab depth at the outer end, DEPTHL, may have already been increased because of the increase in the wall dimension, as indicated by step "a".

222. The procedure for the initial "a" increase in the outer thickness of the base slab is as follows. If the wall thickness in the outer wall is increased during the design cycles for wall strength, then the initial value of DEPTHL will generally be increased by a thickness of about 75 percent of the increase in wall thickness. However, the increase will be limited such that the initial estimate of the outer base slab thickness does not exceed twice the value of the minimum input value. Also, the initial guess for the

outer base slab thickness will not be increased if the wall thickness is less than the input minimum value of DEPTHLI.

223. If uplift is not satisfactory, the value of WHEELL is incremented using the modified half-interval procedure as described earlier subject to the upper limit of WHEELLM. If the uplift requirement is still not satisfied, the base slab is incremented uniformly in 1-in. increments as shown in step "2" until the uplift criteria are satisfied, DEPTHL reaches twice DEPTHLI, or DEPTHC reaches twice DEPTHCI.

224. During the uplift iteration for basins or channels, the effects of anchors are considered. The anchors were described earlier in the investigation mode. The design considering anchors is somewhat limited, because the number and capacity of the anchors must already have been input. Thus, the designer must have already anticipated the need for the anchors prior to the design run. It is likely that the designer would first attempt a solution without the anchors, decide that they were needed, and then do a revised run including the anchors. The iterative design procedure is identical in every respect whether or not anchors are used. However, the maximum capacity of all the anchors is included in computing the factor of safety for uplift. The user should refer to the earlier discussion of maximum anchor force in the investigation mode.

Checks of Bearing Pressure

225. Bearing pressure is checked prior to the frame solution for the empirical foundation option and afterwards for the beam on elastic foundation option. However, the foundation dimensions are not revised if the bearing criteria are not satisfied. The factors of safety concerning bearing are simply reported, and a message is output if the required factor of safety is not achieved. Bearing is seldom a problem for U-frame structures, and the iterative scheme to eliminate the bearing overstress would make the program unduly more complicated. Also, it should be noted that as the iterations for other criteria occur, the status of the bearing check will change and be duly reported.

Design of Base Slab for WSD or SD Criteria

226. Next, the foundation variables are increased as appropriate until stress or strength criteria are satisfied. Since the slab has possibly already been incremented in size because of wall thickness increases or the need to help satisfy uplift, the simple iteration procedure is used in incrementing the slab thicknesses for stress. The slab iteration involves the most recalculations of any of the design steps because the entire solution including the frame analysis is repeated during each iteration. Thus, the program user is warned that inputing an initial value of slab thickness greatly thinner than the walls could cause excessive computer costs. As discussed for the walls, the amount of steel permitted by the user may also influence the size of the section selected by the program.

227. Figure 52 shows the base slab dimensions which are incremented to satisfy the stress or strength criteria for basins. These variables may have already been increased above their input values during the wall or uplift iterations. The iteration shown as "1" is done if the overstress location occurs in the heel portion. The value of DHEE1l may not exceed DEPTHS. The "2" iteration is done either if the overstress occurs in the slab portion or if the "1" iteration is not sufficient for the heel section. DEPTHS may not exceed twice DEPTHSI. No stress or strength checks are made in the "rigid" block under the walls.

228. The iterative scheme to satisfy stress or strength criteria for the channel base slab is shown in Figure 54. The procedure may increment the heel depth, DEPTHL, or the center slab depth, DEPTHC, separately or together according to the following criteria. If DZL is less than 50 percent of DZC, increment DEPTHL; if DZC is less than 50 percent of DZL, increment DEPTHC; if both cases are not true, then increment both DEPTHL and DEPTHC. DZL is the distance from the extreme left edge of the base slab to the location of the overstress, and DZC is the distance from the overstress to the location of the central depth. The user is reminded that the program has an option to output the intermediate iteration steps and that exercising this option may be helpful in understanding the iteration process.

229. The critical sections at which stress or strength criteria are checked in the base slab are the same for both basins and channels except that heels are not checked for channels. For both the basins and channels the

stress or strength criteria are checked internally in the slab at the tenth points. The shear check is made at the face of the walls rather than some distance away since the wall support for the slab is not a well-defined condition for being a tension or compression support. The critical section where stress or strength criteria are checked for the basin heel is at the face of the wall.

230. The checks on shear are made initially with the depth DSH computed assuming the maximum number of steel layers are acting. However, if the shear is critical and flexure is not, this solution is slightly conservative. Thus, for this case, an approximate solution is made to find the required area of steel that is used to compute the value of DSH with which to recheck shear. The user who elects to output the design variable iterations may occasionally see more than one value of the shear stress ratio output for a particular load case (the first value greater than one and a subsequent value less than one). This result indicates that the procedure just described allowed the trial section to satisfy the shear requirement. A similar adjustment based on the required area of steel b ng less than the maximum input by the user is made for the DBAL/D ratio whe using the WSD option and in the ductility ratio for the SD option.

231. During the iterative process, the members are sized such that they ensure the appropriate stress or strength criteria will be satisfied with an amount of steel less than the maximum prescribed by the user or less than that to make the DBAL/D ratio or the ductility ratio equal to one. If any the criteria cannot be satisfied, the user has the option to get the complete output of the results for the detailed study before trying another design. Such outputs contain appropriate warnings when any criteria are not satisfied.

Design Mode Output

232. The output file will contain all the original input values of the design variables and the final incremented values. The final values are clearly distinguished to reduce the possibility of the smaller initial value being accidentally mistaken for the final value. Pressures and member forces are output generally for the analysis mode, except this output and the detailed output described below is limited to members on the left side of the U-frame.

233. After all iterations are completed the final steel requirements are computed as described subsequently and given at the tenth points for all members except heels. Basin heels will have required areas listed at midpoint and end adjacent to the wall. Channel heels will not show any steel requirements. Walls with breaks will also have the areas required at the breaks output.

234. After all areas of steel are found and stored for both sides of a section if needed, the final stresses or the section strength and ductility ratios are computed using these areas and output by load case. If a reversal of the moment at a section requires tension steel on both faces, there would in fact be some compression steel. However, compression steel is not considered in computing the final stresses or making the final strength checks. The only case in which compression steel is taken into account is in the investigation mode. The steel required on the nominal compression face is of course considered in computing stresses for cases with significant axial tension.

235. Because of the iterations involved in both the design and investigation equations using the WSD option, some stresses may be nominally higher than their allowable values. The final stresses are printed for each load case, and if any flexural stress exceeds one-half percent over the corresponding allowable value, a warning message is printed.

236. The procedure used for the SD module should ensure that the final strength and ductility ratios for axial-flexural effects are all less than or equal to one. However, if any of these values exceed 1.005 at the output points, a warning message will be output.

237. The shear stress or strength ratios output for the walls may exceed 1.0 at the base because the wall is usually sized for shear at a distance DSH above the base slab. As usual, the user of any complex design program should thoroughly review the output.

Steel Selection Using WSD Option

238. In the WSD option, the selection of steel is made after the sections have been reviewed and found to satisfy all allowable stress criteria with the steel less than or equal to the maximum amount permitted by the user. Two procedures are followed depending on the number of layers of steel. If one layer of steel is satisfactory at a section, the following simple

procedure is followed. Figure 55a shows the section with a single layer of steel and corresponding variables. The axial force, RN, and moment, RM, used in equations are those at the compression face, as in the investigation procedure.

239. First, it is desirable that the steel stress be at its full allowable value at the critical sections. The minimum depth to ensure this condition, DBAL, is established as follows. Assuming that FC = FCA and FS = FSA, similar triangles (Figure 55b), gives

$$K = FCA/(FCA + FSA/RNS)$$

Summing moments about the tension steel yields

$$RN*D + RM = 0.5*FCA*B*KD*(D-KD/3)$$

Let

$$J = K - 1/3$$

and

$$ALPB = FCA*B*K*J$$

Then solving the quadratic gives

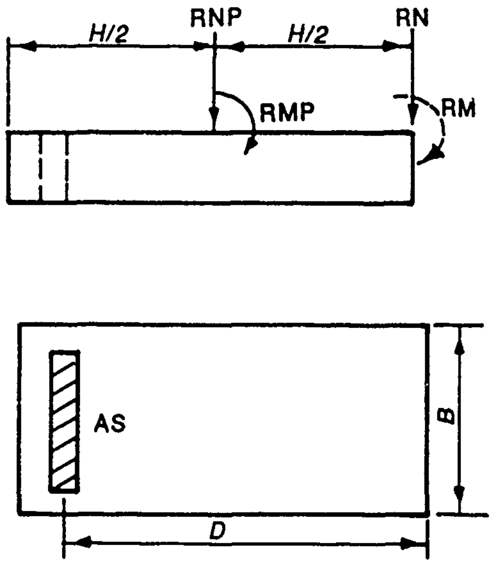
$$DBAL = D = [RN + (RN^2 + 2*ALPB*RM)^{0.5}]/ALPB$$

240. Now that the steel is known to be at its allowable value, the required area of tension steel for flexure can be established as follows. Summing moments about the compression face, assuming FS = FSA, and letting Y = KD yield

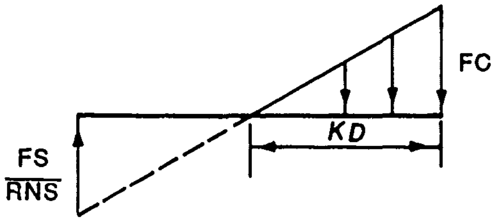
$$RM = AS*FSA*D - B*Y^2*Y*FSA/[6*RNS*(D-Y)]$$

Solving for AS gives

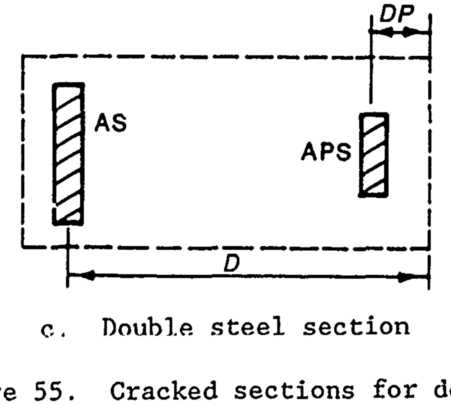
$$AS = \left\{RM + B*Y^3*FSA/[6*RNS*(D-Y)]\right\}/(FSA*D)$$



a. Cross section and forces



b. Stresses on cracked section



c. Double steel section

Figure 55. Cracked sections for design

In order to solve this equation, the value of Y must be known. Summing forces yields

$$RN = 0.5*FSA*Y^2/RNS/(D-Y) - FSA*AS$$

Simplifying and solving the quadratic results in

$$Y/D = -BETA + (BETA^2 + 2*BETA)^{0.5}$$

where

$$BETA = RNS*(RN + FSA*AS)/(FSA*B*D)$$

The equations for AS and Y can be solved iteratively; the initial value for Y = KD is taken as the value for balanced conditions.

241. No solution exists for a negative value of $BETA^2 + 2*BETA$. However, this solution corresponds to a situation of a large axial tension force. The above solution was developed assuming some compression in the concrete. For cases of large axial tension, the value of Y approaches zero and can become negative. The value of axial tension that causes a zero value of Y depends on the eccentricity of the axial force relative to the depth of the section. However, it is not necessary to compute this value since the solution above will fail at precisely this value of axial tension.

242. For negative values of Y, the above solution is invalid since it would give a tension in the concrete. For this condition, the section can be treated as a steel beam as shown in Figure 55c. A steel area, APS, is required on the nominal compression side. Summing moments, then forces, and assuming both areas of steel acting at their full allowable values yield

$$AS = (RN*DP + RM)/[FSA*(D-DP)]$$

and

$$APS = -RN/FSA - AS$$

243. In the above procedures, it was assumed that there was only a single layer of steel or a single layer in each face for the solution for large values of axial tension. Thus, the section is checked to see if the

steel can be provided using only a single layer. If not, the required area of steel is found by a half-interval solution using the area of a single layer as a lower limit and the maximum amount permitted by the user as an upper limit. In this iterative process, the investigation equations described earlier are used to see that the allowable stress criteria are satisfied.

Steel Selection by SD Option

244. In the SD option, the selection of steel is made after the sections have been reviewed and found to satisfy all strength and ductility criteria with the steel less than or equal to the maximum amount permitted by the user. The steel is selected using the half-interval iterative procedure with the lower limit on the area of steel set equal to zero and the upper limit set equal to the maximum amount permitted by the user. All the appropriate strength and ductility criteria described earlier are checked until the area of steel is found within 0.01 sq in.

245. For the usual case of axial compression, only steel on the tension side is assumed. However, for members in tension, a certain ratio of compression to tension steel is assumed. The ratio of compression steel to tension steel is found based on equilibrium for the case of steel on both sides of the section being yielded and the axial force Pu being in tension. Once this ratio is computed, the half-interval procedure proceeds as usual with the steel on both sides of the face incremented until a solution is found that satisfies the strength and ductility criteria.

PART VII: TERMINAL EXECUTION OF PROGRAM

246. The program executes in a terminal control mode. The users may prepare a data file in advance or prepare the data file with an on-line editor which will guide the user in preparing data by only asking for the data required for a particular problem. For example, once the user specifies that the U-frame has only one bay, the on-line editor will only prompt for input related to single bay U-frames. However, users should have read this report and will occasionally need to refer to the appropriate input guide and the associated sketches even if preparing the data file with the aid of the on-line editor. Beginning users are strongly urged to utilize the on-line editor to prepare their input files.

247. Once the data file is prepared, it may be displayed, edited, saved, and executed during the terminal run. Thus, the on-line editor could be used to create several data files during one program run, and these files saved for later execution. Likewise, output obtained may be viewed and/or stored for later printing. A plot file may be prepared to be used later with the plotting program CUFRMP which uses the Corps Graphics Compatibility System 2D (GCS2D) (US Army Engineer Waterways Experiment Station and West Point Military Academy 1982).

Creating and Modifying Data Files Using On-line Editor

248. The on-line editor portion of the program which displays the prompts for editing and creating data is very user friendly. Input is requested by section, using the sections numbers found in the appropriate input guide. However, input is not requested for sections which are not required for the user's particular problem.

249. When a line of input is requested for a section, the editor displays the variable description as well as the program variable name. Values are input on the line below the variable names and must be input in order with one or more spaces placed between values. If a value is not placed on the input line for each variable or if too many values are placed on the input line, the editor will ignore the values and redisplay the variable names when the return key is struck.

250. When editing an existing file, the editor asks the user to decide whether or not to modify each input section one by one, ignoring redundant

sections. A "No" response will move the editor to the next required section. A "Yes" response will prompt the editor to display the required variables with the variable descriptions, variable names, and the current value of each variable. A carriage return by the user is an indication of acceptance of all the current values, and the editor moves to the next required line of input variables.

251. The user may accept the current value of any variable within the line by placing an "S" (for same) in the appropriate space. New values for an individual variable may be input by placing the new value in the appropriate space. For example, for a data line with five variables required, the user might respond

2 s s 15.53 eMP

This input would keep the second and third variables at their same or existing value and redefine the first, fourth, and fifth variables. Floating point data such as a dimension of 15.53 must be entered with the decimal point, but scientific notation is not permitted. However, the decimal point is optional for whole floating point numbers. Integer data such as the number of EM-like load cases should be entered without a decimal point. Key words such as "EMP" are input without quotes and may be upper or lowercase.

252. It is generally a good idea to input the data sections in numerical order. However, an option is provided such that the experienced user can move directly to a particular data section with the on-line editor. When prompted for a "Yes" or "No" response regarding modifying a particular section, the user may respond "GJ," where J is any integer from 1 to 14. The G should be followed by the value of J without any spaces. This response will cause the on-line editor to move to section J for data modification. This option to move to a particular section is very convenient when only one or two sections need to be modified. However, the user is warned that if a section is skipped, the program will not request any data for that section, even if other changes in the data require some change in the skipped section. Users may also elect to exit the on-line editor anytime when prompted for a "Yes"/"No" response to modify a particular section by responding "Q" for quit.

253. Finally, the users are reminded that there will be no prompting for variables that are not needed by the program for a particular problem.

Thus, while the input guide may describe eight values of input for a line in the most general case, if only five values are needed because of the options selected, the users will only be prompted for those five values (i. e. USE THE ON-LINE EDITOR!).

Program Execution

254. Figure 56 presents a summary flowchart of the terminal execution of the program. The flowchart shows that an early response requested from the user is to indicate whether or not an existing data file is to be input. Such responses will be either "YES" or "NO" ("YE", "Y", and "N" are also acceptable responses). If a previously prepared data file is to be used, then the name of the data file must of course be input. If the user responds "YES," indicating an old file is to be input, the program will read the data file named and prompt for another "YES"/"NO" response indicating whether or not the data file is to be displayed on the terminal. If the data file is displayed at this time, it will be shown as a raw data file without any accompanying headings.

255. Next, as seen in the flowchart, the user will be asked to indicate whether it is necessary to modify the data file as input or if a new data file is to be created. If the "MOD" option is selected, then the user will be given the necessary prompts by the on-line editor to edit the existing data file. If the "CRE" option is selected, the on-line editor user will provide the prompts to prepare a new data file. The user will be given the option to see a summary of instructions on how to apply the on-line editor if the on-line editor is selected. Then according to the flowchart, the program control returns to the portion where the user is prompted to indicate if the data file should be displayed.

256. Eventually, the user will be satisfied with the data file and respond "NO" to the query on creating or modifying the data file. At that time, the flowchart indicates that the user has the option of storing the data in a permanent data file. Data files that are stored may or may not have line numbers. If line numbers are chosen, they are numbered such that the first two digits of the line number are the data section number.

257. Next, assuming an investigation problem is being run, the decision is made by the user whether or not the data file now active in the program is

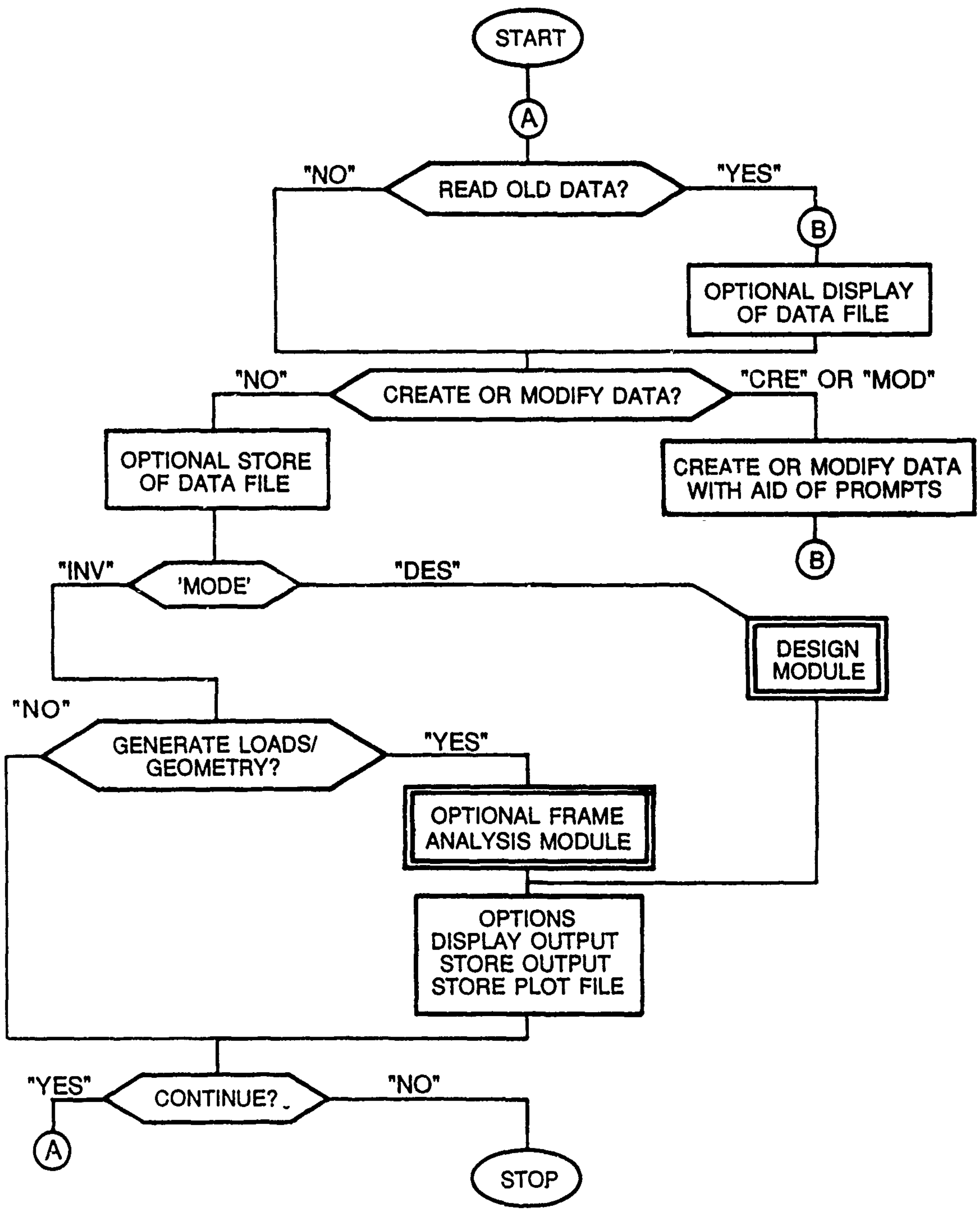


Figure 56. Summary flowchart for terminal execution

ready to be executed. The execution is broken up into two phases. First, the loads are generated and the frame geometry defined. After the first phase, the user has the option of continuing on to the detailed frame analysis or not. The terminal will display factors of safety and the horizontal equilibrium for the appropriate load cases prior to prompting for the decision on whether or not to do the detailed frame analysis.

258. At this point, an output file is now created with either the results of the preliminary analysis or the complete analysis. It also contains the input with appropriate headings. This output file may be displayed at the terminal and/or stored for future listing. Also, the option is provided for storing the necessary plot data, such that the user may obtain plotted output in a later execution of the CUFRMP graphics program.

259. Now the user may stop the run or continue the program. If the program is continued, the user may modify the existing data file in the program, create a new one, or input any other existing data file. This flexibility allows the user to perform a variety of investigations varying important parameters or to iterate to a design that has acceptable output very quickly.

260. If the run is made using the design option, the flow is slightly different. After the data file is ready, the program branches to the DESIGN MODULE as shown on the flowchart. Here the user is asked whether or not the design should be continued. Assuming the user continues, the program follows the design algorithm flowchart previously discussed until the output and plot files are prepared. From that point on, the flow is identical to the investigation mode. During the design, the intermediate values of the design variables and the corresponding stress or strength ratios can be displayed at the terminal if requested by the user.

Semibatch Mode

261. The above described procedure gives the user maximum control over the program at the cost of a few minutes of time. However, after several runs of the same problem or runs of several problems with previously prepared data files, a semibatch mode is available to reduce the interaction time. If the semibatch mode is selected, the user is only required to give the name of the input file, whether or not it is a line-numbered file, and whether or not the

response is to continue on to a new problem. Depending on the type of terminal being used, it may even be possible to stack a series of problems by inputing these three responses to a series of problems on the terminal screen during pauses in the response from the host computer.

262. If the semibatch mode is selected, then the user must be prepared to accept the consequences of loss of control of the process. In the semibatch mode, the program generally takes the more complete, longer, and more costly of the options that would be available to the user in the terminal control mode. However, intermediate values of the stress or strengths ratio are not output for the semibatch mode.

File Conventions

263. The data file, output file, and plot file are all given unique file names of up to six characters by the user. The data file will in fact contain all of these names. It should be noted that while the output file will always contain the information in the data file, it is still desirable to maintain the data file for documentation purposes or possible later modification. Also, the data file that is used is the one that exists at the time of the execution of the solution. Thus, it is possible to execute the program with a data file that is not stored as a permanent file.

264. Experienced users may wish to prepare the data files in advance of the program execution using their own editor. Such files must be in the American Standard Code for Information Interchange (ASCII) format and optionally may have line numbers of up to six integers at the extreme left of the file. The data file is a free format with input items either numbers or alphanumeric data. The items are separated by one or more spaces. Floating point and integer data should be typed as described earlier for the on-line editor.

265. The data are structured sequentially in sections and lines. The sections are numbered as indicated in the input guide. Each section asks for a certain number of lines of data, and each line should contain a certain number of data items. However, as indicated in the input guides, certain lines and data items on lines are omitted depending on the options selected. As a data file is read, it is checked for the correct number of items in each sequential line. If a line has an incorrect number of items, a message is

displayed indicating the section number of the erroneous line, and the program terminates to allow the user to correct the data file. When entering the input directly with the on-line editor, if the wrong number of items are input for a line, the user is reprompted for the data line.

266. Since free format input is used, it is possible that some very small values could be input and used in the program. However, the input file and the output file contain only a finite number of places after the decimal point. Thus, a very small input number could conceivably be lost in the input and output files. For writing most input quantities to the input or output file, the program generally uses three places after the decimal point in order to represent all reasonable data to satisfactory accuracy.

267. A limited number of checks are made on the acceptability of program data by the on-line editor. For instance, water elevations are not permitted to exceed the height of an adjacent wall. The data checks are generally made just prior to the solution of the program. If any unacceptable data are encountered, the user will be allowed to either modify the data using the on-line editor, store the data file for future modification, or terminate the run. However, it is not possible to provide checks for all data that might be incorrect, and it is obviously impossible to ensure that the input data will correctly model the user's given problem when applied to the program. Thus, the program user must thoroughly review the program output to ensure that the data selected were appropriate for the particular U-frame.

Macro Flowchart/File Management

268. CUFRBC is a rather lengthy FORTRAN program consisting of nine modular first-level and five second-level Program Overlays. Figure 57 shows a listing of the Overlay Programs and their Subroutines. The zero level overlay contains the main program, CUFRBC. The main program controls the solution and performs some simple preprocessing tasks. It also calls all the major design module Subroutines, which are contained in the zero-level overlay.

269. Program WEDGE does the active and passive solutions as well as the empirical wall pressure calculations. Program WFILE writes the output to either the terminal or an output file and computes concrete and steel stresses or section strength and ductility ratios. Program GEOM creates the geometric frame model from input data. Program WATER calculates hydraulic pressures and

```
C - ********************************************************************
COMMENT - LIST OF PROGRAM OVERLAYS AND SUBROUTINES
C - ********************************************************************
C------OVERLAY (0,0) = MAIN PROGRAM CUFRBC
C------SUBROUTINES
C      DESIGNW
C      DESIGNU
C      CHECKB
C      DESIGNF
C      INTEGER(DEC,DECI,STEP)
C      COVERS (COVT,COVB,IUFS)
C      BAR (NBAR8,DIAMBT,AREABT,SPBAR)
C      ASTRESS(B,H,NLTS,NLC,RNS,RNPS,DS,DPS,AS,APS,RMP,RNP,Y,FC,FS,FPS
C             ,FCT,NITNEW)
C      POAPMCI (B,DS,RNPS,AS,APS,DPS,RMP,RNP,FC,FPS,FS,H,NLTS,NLC,FCT)
C      NEWTON (CO,DCO,ROOT,TOL,ZER,MAXIT,IDEGR,NITNEW)
C      TENSION (AS,APS,DS,DPS,RN,RM,FS,FPS,NLT,NLC)
C      ALLSHR (AXF,AG,VCAB,SQFPC)
C      ASBAS (B,D,H,RNS,FCA,FSA,RMP,RNP,AS,Y,DP,APS)
C      DBALS(COVERT,COVERC,CCLAY,THP,DIAMBT,AREABT,NLTSF,SUM,AST,
C            B,DBAL,RMP,RNP,FCA,FSA,RNS)
C      SDINV (FPC,EPM,BETAM,FCRD,PMAXFD,PHIAD,PHIFD,PHIS,BET1
C            ECM,NCODED,THP,NLTS,NLC,DTS,DPS,ASD,APS,RMP,RNP,FYD,AG,
C            PBRAT,SDLF,AST,APST,XSECT,AXF,DSH,SHP,ICNTRL,IMODE,IFAIL)
C --------------------------------------------------------------------
C THE NEXT 5 SUBROUTINES AND FUNCTION SLOPE CAME FROM KOE PROG.
C                          CSTR(X0066S)
C      ARRAY
C      PHIFAC
C      DIAG
C      CHECK
C      USED (X,Y,XP,YP,PCNT,NZONE)
C--------- FUNCTION - SLOPE(XX,YY,IFO)
C --------------------------------------------------------------------
C      USSHR (AXF,AG,DSH,SQFPC,PHIS,SHP,SDLF,VCN,VCRAT)
C      DATA55
C      LOAD
C      ADDONE (FUN,NR,NRM)
C      HEAD
C      CONVERT
C      UNTIL (ILOOK,IALP)
C      FACWED(III,IBTYPE,N)
C      SPD
C      INTERP (XSECT,TH,FUN,VFUN,NUPTS,IBS,IUFS,N)
C      EDCFR(IVAL)
C      IALPST(IALPN,IALP)
C      YESNO(INPUT,IOUTPUT)
C - ********************************************************************
```

Figure 57. Overlay and subroutine listing (Sheet 1 of 3)

```
C -----OVERLAY (2,0) = PROGRAM WEDGE
C------SUBROUTINES
C       MAXHAF(XLS,XRS,XM,FM,DELTA,NIT)
C       MINHAF(XLS,XRS,XM,FM,DELTA,NIT)
C       FUNX(X,FX,ICON)
C       EMP(IW)
C       SPRESS (HWAT,HSOIL,UWD,UWS,GAMAW,PRESS)
C - *****************************************************************
C -----OVERLAY (3,0) = PROGRAM WFILE
C------SUBROUTINES
C       LDCAS(STRESS,LC,IDLC,NEM,NLOCT,IUFS,IWRITE)
C       STRSUM(III,IUFS,LOC,VC,VCM,LCV,IPOSM,FC,FS1,FPS1,FCT,FCBM,FCTM,
C       LCFCBM,LCFCTM,FSBM,LCFSBM,FSPBM,LCFSPBM,FSTM,LCFSTM,FPSTM,
C       LCFPSTM,FCTBM,LCFCTBM,FCTTM,LCFCTTM)
C - *****************************************************************
C -----OVERLAY (4,0) = PROGRAM GEOM
C------SUBROUTINES
C       ADDCOR (X,Y,NM,J1,J2,NCALL)
C       SECT
C       STIFF
C - *****************************************************************
C -----OVERLAY (5,0) = PROGRAM WATER
C------SUBROUTINES
C       SPECLD (NM,INS,NMLS)
C       DISTCON (DIST,X,NM,LDME,XLEN)
C - *****************************************************************
C -----OVERLAY (6,0) = PROGRAM DATA
C------SUBROUTINES
C       STRIPS,RETURN,(R88)
C       ALINE(LINE,LINEA)
C       KEYCK(IGI,II,KEYERR)
C - *****************************************************************
C -----OVERLAY (6,1) = PROGRAM DATAIN
C -----OVERLAY (6,2) = PROGRAM DATAPRT
C -----OVERLAY (6,3) = PROGRAM DATAMOD
C -----SUBROUTINES
C       IGOS(IMOD2,IGO)
C -----OVERLAY (6,4) = PROGRAM DATASTR
C -----OVERLAY (6,5) = PROGRAM DATACK
C - *****************************************************************
C -----OVERLAY (7,0) = PROGRAM UFRDAT
C -----SUBROUTINES
C       SDPAR (FC,EMAX,BM,FCR,PMAXF,PHIA,PHIF,PHIS,B1,EC,NCODE)
C - *****************************************************************
C -----OVERLAY (10,0) = PROGRAM DESIGNS
C - *****************************************************************
C -----OVERLAY (11,0) = PROGRAM EMPFD
C -----SUBROUTINES
C       PINT(RP,DX,XX,RPYN)
C       FSPECLD(NM,NMLS,N3P2,NLDM)
C       DISCON (DIST,X,NM,LDME,XLEN)
```

Figure 57. (Sheet 2 of 3)

```
C - ********************************************************************
C -----OVERLAY (1,0) = PROGRAM MAIN55
C------SUBROUTINES
C       FAE (DELTA, TAU1, TAU2, I, TT, BM1,BM2)
C       FORMST (RM, RO, W, SL, SU, SMMT, L1, L3, L4, L6)
C       DISCST (NC51T, NCDST, ZL, L1)
C       LINSTF (STL,STR,ST,L1)
C       LINLD (QL, QR, Q, L1)
C       CONLD (QI, Z, QO, L1)
C       MEMENI (W,FMMI,L6)
C       GRIP2A (C,B,X,SL,SU,L4,L6,M)
C       FSUB1 (RM,RO,W,L4,L6,SU,M)
C       FSUB21 (SU4, FF, L4, IHB)
C       FSUB22 (SU, FF, L4)
C       ELEMST (I)
C       ELEMFL (DX1,DY1,DZ1,DX2,DY2,DZ2,I,U1T,V1T,W1T,U2T,V2T,W2T)
C       MEMLOC
C       RDMST
C       JNTDAT
C       RDMLD
C       NLSSJR (SXYZ,DXYZ,SQXYZ,L1,NSL,NSR,QML,WML,QMR,WMR,XLT,XRT)
C       REVERSE(A,N)
C       MEMEND (FMM)
C       NLSS (L1)
C       FORMLD (RM, RO, W, SL, SU, FOMT, L1, L4, L6, JJ)
C       DISCLD (L1)
C       MATM33 (AA, BB, CC)
C       MATM31 (AA, B, C)
C       JNTSPR (SJX, SJY, SJZ, QJX, QJY, QJZ, JTN)
C       CURVE (QQ, WW, WJ, NPT, ISYM, QJ, S2, KOFFC)
C       SANGLE (SA,SXT,SYT)
C       MEMSOL (JJ,RM,RO,W,SU,L1,L4,L6)
C       ADJTER (F1M, F2M, J1, J2, DC1, DC2)
C       JTCORD
C       PRINT9 (AN2,N PROB,RM,RO,W,SU,L1,L4,L6)
C       ITCONT
C       FRAM55 (RM, RO, W, SU, L1, L4, L6)
C - ********************************************************************
```

Figure 57. (Sheet 3 of 3)

prepares special loads for the frame analysis module.

270. Program DATA is the on-line editor driver picking the appropriate second-level overlay to readIN an existing data file, PRinT an existing data file, MODify an existing data file (or create a new one), SToRe a data file, or ChecK the existing data for consistency. Program UFRDAT converts the specific basin or channel data to general U-frame data. Program DESIGNS selects the steel areas. Program EMPFD does the empirical foundation solution. Program MAIN55 (Program MAIN55 is generally referred to as FRAME55) is the frame analysis module and does the detailed analysis using the frame model, considering the soil-structure interaction features required.

271. Figure 58 shows a macro flowchart of the program emphasizing the uses of the various permanent and scratch files. The first phase of the program consists of the input and editing of data for either a basin or channel structure. Data may be read off a permanent file or the screen. If a permanent file is used, it is assigned to unit 10 with the internal name of INFILE. The actual file name is input by the user. Terminal input for data or program control is off of unit 5. Scratch units 1 and 2 are used to rewrite the free-form input data as formatted data before being actually input to the program. Terminal prompts and output are written to unit 6.

272. The basin or channel data are converted into common data variables for a general N-wall U-frame structure. Next, the U-frame data are written onto a scratch file, unit 8. The data are written onto the scratch file in the prescribed format for input to the frame analysis module which is program FRAME55. All input to FRAME55 is via this data file.

273. Next, FRAME55 does the actual frame analysis for joint displacement, member displacements, and forces. Scratch units 1, 2, and 3 are utilized in this program. The output of FRAME55 is written to a scratch file, unit 9. This output is essentially the normal annotated output of the existing program FRAME55. The program then reads the FRAME55 output from unit 9 and converts it into the output desired for the U-frame channel or basin. The U-frame output may be displayed on unit 6 (terminal) or stored on unit 11, which is assigned the name of the output file specified by the user. Plot data may be stored on unit 7, which is assigned the name of the Plot data file specified by the user.

274. There is an option to keep units 8 and 9 as permanent files. To activate this option the first four letters in the second line of the Header

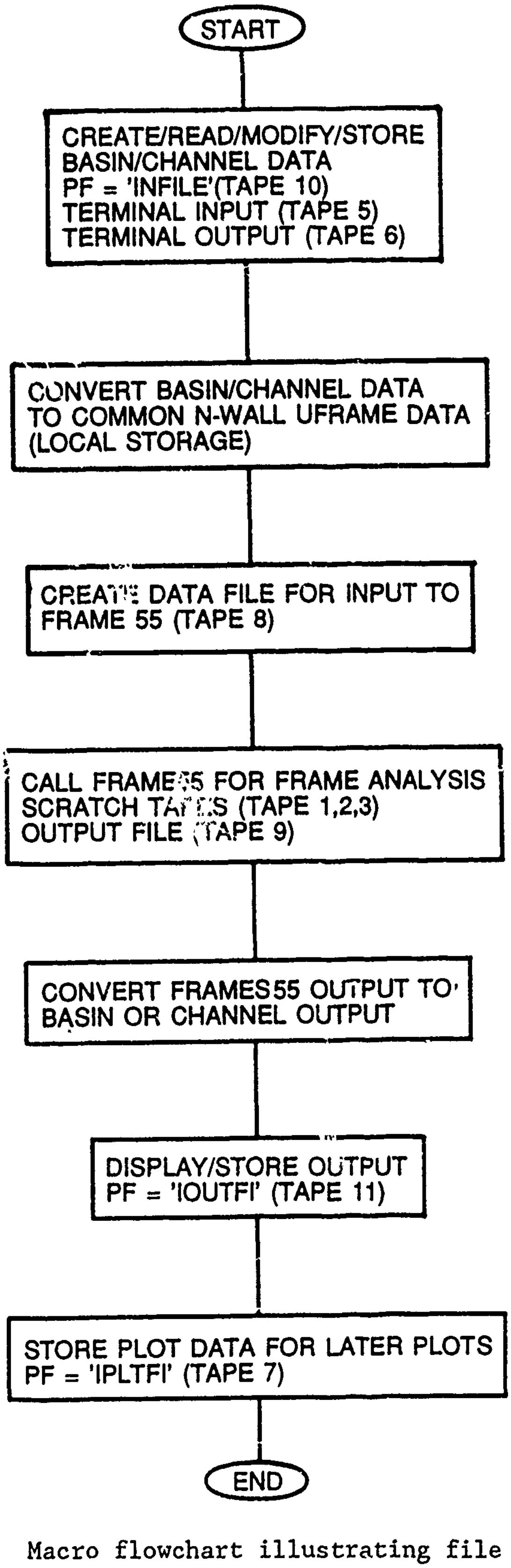


Figure 58. Macro flowchart illustrating file management

should be 'DBUG.' Units 8 and 9 contain the standard input and output for program FRAME55 and would only be required for a problem needing extensive debugging. In addition, a number of temporary outputs will be activated in the DBUG option that will show the internal position at key locations within the program. The permanent file names assigned to units 8 and 9 are 'S8' and 'S9.' Note that these permanent files are not created unless the DBUG option is activated.

275. Another hidden option allows a study of the time spent in the various programs and subroutines in the program. To activate this option, set the first four letters in the second line of the header to 'TIME.' In addition to the primary frame solution by FRAME55, FRAME55 is also used for a preliminary analysis to compute the summation of vertical and horizontal forces and moments on the U-frame when the empirical foundation option is used.

276. The program CUFRBC has been developed on the Cybernet Interactive System available through the US Army Corps of Engineers, WES. However, the programming is the standard FORTRAN IV and should be easily converted to other systems. File assignments are handled by using Subroutine PFGET and PFREPLC available through the attachment of WES library IOLIB2.

Plotting Program CUFRMP

277. Because of the large size of the program CUFRBC, it was decided to have a separate program for plotting the results. ing execution of CUFRBC, the user may store request that the results needed for plotting be stored on a permanent file. Then plotted output may be obtained at any later time through the use of the Fortran program CUFRMP.

278. CUFRMP is a Fortran program that allows the user to obtain graphical output on any of the hardware supported by the GCS2D system. The entire program is interactive, and the user merely responds to simple questions concerning what types of output are desired and for which load cases the output is needed. Detailed descriptions of the output available for plotting were given earlier in this report. The types of output available are:

a. U-frame geometry including soil and water elevations.

b. Individual wall pressure plots.

c. Base slab pressure plots.

d. Member force and deflection plots.

e. Plots of required areas of flexural steel.

279. At the start of CUFRMP, the user is prompted for the name of the file containing the plot data, which was input in Section 2 of the data.

REFERENCES

American Concrete Institute. 1956. "Building Code Requirements for Reinforced Concrete," ACI 318-56, Detroit, Mich.

__________. 1983. "Building Code Requirements for Reinforced Concrete," ACI 318-83, Detroit, Mich.

Haliburton, T. A. 1972 (Feb). "Soil Structure Interaction," Technical Publication Number 14, Oklahoma State University, Stillwater, Okla.

Hamby, C. C., and Price, W. A. 1984 (Sep). "User's Guide for Concrete Strength Investigation and Design (CSTR)," Instruction Report K-84-9, US Army Engineer Waterways Experiment Station, Vicksburg, Miss.

Hays, C. O. 1971. "A Nonlinear Analysis of Statically Loaded Plane Frames Using A Discrete Element Model," Ph.D. Dissertation, The University of Texas, Austin, Tex.

__________. 1982 (Aug). "FRAME 54, Nonlinear Analysis of Statically Loaded Plane Frames Using a Discrete Element Model, A User's Guide," University of Florida, Gainesville, Fla.

Hays, C. O., and Matlock, H. M. 1973 (Oct). "Nonlinear Discrete Element Analysis of Frames," Journal of the Structural Division of the American Society of Civil Engineers.

Hays, C. O., and Santhanam, T. K. 1979 (July). "Inelastic Section Response by Tangent Stiffness Method," Journal of the Structural Division of the American Society of Civil Engineers.

Headquarters, Department of the Army. 1961a (May). "Structural Design of Spillways and Outlet Works," Engineering Manual 1110-2-2400, Washington, DC.

__________. 1961b (May). "Retaining Walls," Engineering Manual 1110-2-2502, Washington, DC.

__________. 1981 (June). "Sliding Stability For Concrete Structure," Engineer Technical Letter 1110-2-256.

__________. 1988 (Mar). "Strength Design Criteria for Reinforced Concrete Hydraulic Structures," Engineer Technical Letter 1110-2-312.

Matlock, H. M., and Haliburtion, T. M. 1966 (Sep). "A Finite Element Solution for Linearly Elastic Beam Columns," Research Report 56-1, Center for Highway Research, University of Texas, Austin, Tex.

Shushkewich, K. W. 1983 (Nov - Dec). "Simplified Cracked Section Analysis," Journal of the American Concrete Institute, Detroit, Mich.

Taylor, T. P., and Matlock, H. M. 1968. "A Finite Element Method of Analysis for Composite Reams," Research Report 56-10, Center for Highway Research, University of Texas, Austin, Tex.

US Army Engineer Waterways Experiment Station and West Point Military Academy. 1982 (Revised). "Graphics Compatibility System (GCS) Programmer's Reference Manual," Instruction Report K-82-3, Vicksburg, Miss.

WATERWAYS EXPERIMENT STATION REPORTS PUBLISHED UNDER THE COMPUTER-AIDED STRUCTURAL ENGINEERING (CASE) PROJECT

	Title	Date
Technical Report K-78-1	List of Computer Programs for Computer-Aided Structural Engineering	Feb 1978
Instruction Report O-79-2	User's Guide. Computer Program with Interactive Graphics for Analysis of Plane Frame Structures (CFRAME)	Mar 1979
Technical Report K-80-1	Survey of Bridge-Oriented Design Software	Jan 1980
Technical Report K-80-2	Evaluation of Computer Programs for the Design/Analysis of Highway and Railway Bridges	Jan 1980
Instruction Report K-80-1	User's Guide. Computer Program for Design/Review of Curvilinear Conduits/Culverts (CURCON)	Feb 1980
Instruction Report K-80-3	A Three-Dimensional Finite Element Data Edit Program	Mar 1980
Instruction Report K-80-4	A Three-Dimensional Stability Analysis/Design Program (3DSAD)	
	Report 1: General Geometry Module	Jun 1980
	Report 3: General Analysis Module (CGAM)	Jun 1982
	Report 4: Special-Purpose Modules for Dams (CDAMS)	Aug 1983
Instruction Report K-80-6	Basic User's Guide. Computer Program for Design and Analysis of Inverted-T Retaining Walls and Floodwalls (TWDA)	Dec 1980
Instruction Report K-80-7	User's Reference Manual. Computer Program for Design and Analysis of Inverted-T Retaining Walls and Floodwalls (TWDA)	Dec 1980
Technical Report K-80-4	Documentation of Finite Element Analyses	
	Report 1: Longview Outlet Works Conduit	Dec 1980
	Report 2: Anchored Wall Monolith, Bay Springs Lock	Dec 1980
Technical Report K-80-5	Basic Pile Group Behavior	Dec 1980
Instruction Report K-81-2	User's Guide. Computer Program for Design and Analysis of Sheet Pile Walls by Classical Methods (CSHTWAL)	
	Report 1: Computational Processes	Feb 1981
	Report 2: Interactive Graphics Options	Mar 1981
Instruction Report K-81-3	Validation Report. Computer Program for Design and Analysis of Inverted-T Retaining Walls and Floodwalls (TWDA)	Feb 1981
Instruction Report K-81-4	User's Guide. Computer Program for Design and Analysis of Cast-in-Place Tunnel Linings (NEWTUN)	Mar 1981
Instruction Report K-81-6	User's Guide. Computer Program for Optimum Nonlinear Dynamic Design of Reinforced Concrete Slabs Under Blast Loading (CBARCS)	Mar 1981
Instruction Report K-81-7	User's Guide. Computer Program for Design or Investigation of Orthogonal Culverts (CORTCUL)	Mar 1981
Instruction Report K-81-9	User's Guide. Computer Program for Three-Dimensional Analysis of Building Systems (CTABS80)	Aug 1981
Technical Report K-81-2	Theoretical Basis for CTABS80. A Computer Program for Three-Dimensional Analysis of Building Systems	Sep 1981
Instruction Report K-82-6	User's Guide. Computer Program for Analysis of Beam-Column Structures with Nonlinear Supports (CBEAMC)	Jun 1982
Instruction Report K-82-7	User's Guide. Computer Program for Bearing Capacity Analysis of Shallow Foundations (CBEAR)	Jun 1982

(Continued)

WATERWAYS EXPERIMENT STATION REPORTS PUBLISHED UNDER THE COMPUTER-AIDED STRUCTURAL ENGINEERING (CASE) PROJECT

(Continued)

	Title	Date
Instruction Report K-83-1	User's Guide. Computer Program With Interactive Graphics for Analysis of Plane Frame Structures (CFRAME)	Jan 1983
Instruction Report K-83-2	User's Guide. Computer Program for Generation of Engineering Geometry (SKETCH)	Jun 1983
Instruction Report K-83-5	User's Guide. Computer Program to Calculate Shear, Moment, and Thrust (CSMT) from Stress Results of a Two-Dimensional Finite Element Analysis	Jul 1983
Technical Report K-83-1	Basic Pile Group Behavior	Sep 1983
Technical Report K-83-3	Reference Manual. Computer Graphics Program for Generation of Engineering Geometry (SKETCH)	Sep 1983
Technical Report K-83-4	Case Study of Six Major General-Purpose Finite Element Programs	Oct 1983
Instruction Report K-84-2	User's Guide. Computer Program for Optimum Dynamic Design of Nonlinear Metal Plates Under Blast Loading (CSDOOR)	Jan 1984
Instruction Report K-84-7	User's Guide. Computer Program for Determining Induced Stresses and Consolidation Settlements (CSETT)	Aug 1984
Instruction Report K-84-8	Seepage Analysis of Confined Flow Problems by the Method of Fragments (CFRAG)	Sep 1984
Instruction Report K-84-11	User's Guide for Computer Program CGFAG, Concrete General Flexure Analysis with Graphics	Sep 1984
Technical Report K-84-3	Computer-Aided Drafting and Design for Corps Structural Engineers	Oct 1984
Technical Report ATC-86-5	Decision Logic Table Formulation of ACI 318-77, Building Code Requirements for Reinforced Concrete for Automated Constraint Processing, Volumes I and II	Jun 1986
Technical Report ITL-87-2	A Case Committee Study of Finite Element Analysis of Concrete Flat Slabs	Jan 1987
Instruction Report ITL-87-1	User's Guide. Computer Program for Two-Dimensional Analysis of U-Frame Structures (CUFRAM)	Apr 1987
Instruction Report ITL-87-2	User's Guide. For Concrete Strength Investigation and Design (CASTR) in Accordance with ACI 318-83	May 1987
Technical Report ITL-87-6	Finite-Element Method Package for Solving Steady-State Seepage Problems	May 1987
Instruction Report ITL-87-3	User's Guide. A Three Dimensional Stability Analysis/Design Program (3DSAD) Module	Jun 1987
	Report 1: Revision 1: General Geometry	Jun 1987
	Report 2: General Loads Module	Sep 1989
	Report 6: Free-Body Module	Sep 1989
Instruction Report ITL-87-4	User's Guide. 2-D Frame Analysis Link Program (LINK2D)	Jun 1987
Technical Report ITL-87-4	Finite Element Studies of a Horizontally Framed Miter Gate	Aug 1987
	Report 1: Initial and Refined Finite Element Models (Phases A, B, and C), Volumes I and II	

(Continued)

WATERWAYS EXPERIMENT STATION REPORTS PUBLISHED UNDER THE COMPUTER-AIDED STRUCTURAL ENGINEERING (CASE) PROJECT

(Continued)

	Title	Date
Technical Report ITL-87-4	Finite Element Studies of a Horizontally Framed Miter Gate Report 2: Simplified Frame Model (Phase D) Report 3: Alternate Configuration Miter Gate Finite Element Studies—Open Section Report 4: Alternate Configuration Miter Gate Finite Element Studies—Closed Sections Report 5: Alternate Configuration Miter Gate Finite Element Studies—Additional Closed Sections Report 6: Elastic Buckling of Girders in Horizontally Framed Miter Gates Report 7: Application and Summary	Aug 1987
Instruction Report GL-87-1	User's Guide. UTEXAS2 Slope-Stability Package, Volume I, User's Manual	Aug 1987
Instruction Report ITL-87-5	Sliding Stability of Concrete Structures (CSLIDE)	Oct 1987
Instruction Report ITL-87-6	Criteria Specifications for and Validation of a Computer Program for the Design or Investigation of Horizontally Framed Miter Gates (CMITER)	Dec 1987
Technical Report ITL-87-8	Procedure for Static Analysis of Gravity Dams Using the Finite Element Method — Phase Ia	Jan 1988
Instruction Report ITL-88-1	User's Guide. Computer Program for Analysis of Planar Grid Structures (CGRID)	Feb 1988
Technical Report ITL-88-1	Development of Design Formulas for Ribbed Mat Foundations on Expansive Soils	Apr 1988
Technical Report ITL-88-2	User's Guide. Pile Group Graphics Display (CPGG) Post-processor to CPGA Program	Apr 1988
Instruction Report ITL-88-2	User's Guide for Design and Investigation of Horizontally Framed Miter Gates (CMITER)	Jun 1988
Instruction Report ITL-88-4	User's Guide for Revised Computer Program to Calculate Shear, Moment, and Thrust (CSMT)	Sep 1988
Instruction Report GL-87-1	User's Guide. UTEXAS2 Slope-Stability Package, Volume II, Theory	Feb 1989
Technical Report ITL-89-3	User's Guide. Pile Group Analysis (CPGA) Computer Group	Jul 1989
Technical Report ITL-89-4	CBASIN--Structural Design of Saint Anthony Falls Stilling Basins According to Corps of Engineers Criteria for Hydraulic Structures; Computer Program X0098	Aug 1989
Technical Report ITL-89-5	CCHAN--Structural Design of Rectangular Channels According to Corps of Engineers Criteria for Hydraulic Structures; Computer Program X0097	Aug 1989
Technical Report ITL-89-6	The Response-Spectrum Dynamic Analysis of Gravity Dams Using the Finite Element Method; Phase II	Aug 1989
Contract Report ITL-89-1	State of the Art on Expert Systems Applications in Design, Construction, and Maintenance of Structures	Sep 1989

(Continued)

WATERWAYS EXPERIMENT STATION REPORTS PUBLISHED UNDER THE COMPUTER-AIDED STRUCTURAL ENGINEERING (CASE) PROJECT

(Continued)

	Title	Date
Instruction Report ITL-90-1	User's Guide. Computer Program for Design and Analysis of Sheet Pile Walls by Classical Methods (CWALSHT)	Feb 1990
Technical Report ITL-90-3	Investigation and Design of U-Frame Structures Using Program CUFRBC Volume A: Program Criteria and Documentation Volume B: User's Guide for Basins Volume C: User's Guide for Channels	May 1990