**US Army Corps
of Engineers**
Waterways Experiment
Station

*Computer-Aided Structural Engineering Project*

# User's Manual — GDAP, Graphics-Based Dam Analysis Program Version 3.25

*by* Yusof Ghanaat
*Quest Structures*

AD-A269 446

DTIC
ELECTE
SEP 16 1993
S
A
D

WES

Approved For Public Release; Distribution Is Unlimited

93 9 15 024

93-21587

Prepared for   Headquarters, U.S. Army Corps of Engineers

The contents of this report are not to be used for advertising, publication, or promotional purposes. Citation of trade names does not constitute an official endorsement or approval of the use of such commercial products.

PRINTED ON RECYCLED PAPER

# User's Manual — GDAP, Graphics-Based Dam Analysis Program Version 3.25

by Yusof Ghanaat

Quest Structures
1900 Powell St.
Suite 210
Emeryville, CA   94608

Accesion For

| NTIS   CRA&I | ☑ |
| DTIC   TAB | ☐ |
| Unannounced | ☐ |
| Justification | |

By
Distribution/

Availability Codes

| Dist | Avail and / or Special |

A-1

DTIC QUALITY INSPECTED 1

**Final report**

Approved for public release; distribution is unlimited

US Army Corps
of Engineers
Waterways Experiment
Station

# TABLE OF CONTENTS

# TABLE OF FIGURES

# PREFACE

This technical report documents the computer program Graphics-based Dam Analysis Program (GDAP) which is used to perform three-dimensional finite element static and dynamic analyses of concrete arch and gravity dams on a desktop computer and to provide graphical pre- and post-processing capabilities. The computer program is an adaptation of the Arch Dam Analysis Program which was developed for the U.S. Bureau of Reclamation. Dr. Yusof Ghanaat (QUEST Structures/consultant) extensively modified and enhanced the original program and developed the proprietary version of the GDAP program. Under contract DACW39-88-C-0055-P003, Dr. Ghanaat adapted and documented the proprietary GDAP program to meet Corps criteria and for U.S. Government use. Funding for the adaptation of the program and documentation was provided by Headquarters, U.S. Army Corps of Engineers (HQUSACE), under the Computer-Aided Structural Engineering (CASE) Project.

This user manual provides a brief theoretical background and model description of the procedures for the static and dynamic analyses of concrete arch dams. An accompanying report, "Theoretical Manual for the Analysis of Arch Dams," was also written by Dr. Ghanaat for the CASE project which describes the analytical procedures employed in GDAP.

This user manual was written under the direction of and is a product of the CASE Arch Dam Task Group. The manual was written by Dr. Ghanaat . Task group members during the development of this manual were:

| | |
|---|---|
| Mr. Byron J. Foster | CESAD-EN (Chairman) |
| Mr. G. Ray Navidi | CEORH-ED |
| Mr. William K. Wigner | CESAJ-EN |
| Mr. Terry W. West | FERC (formerly CESAJ-EN) |
| Mr. Jerry L. Foster | CECW-ED (formerly FERC) |
| Mr. Donald R. Dressler | CECW-ED |
| Mr. H. Wayne Jones | CEWES-IM-DS (Task Group Coordinator) |
| Mr. David A. Dollar | USBR |
| Mr. Larry K. Nuss | USBR |
| Dr. Yusof Ghanaat | Consultant - QUEST Structures |

The work was accomplished under the general supervision of Dr. N. Radhakrishnan, Director, Information Technology Laboratory (ITL), U.S. Army Engineer Waterways Experiment Station (WES), and under the direct supervision of Mr. H. Wayne Jones, Chief, Scientific Engineering and Applications Center (SEAC), Computer-Aided Engineering Division (CAED), ITL, WES. The technical monitor for HQUSACE was Mr. Don Dressler.

At the time of publication of this report, Director of WES was Dr. Robert W. Whalin. Commander was COL. Leonard G. Hassell, EN.

# 1. INTRODUCTION

The graphics-based dam analysis program (GDAP) performs three-dimensional (3-D), finite element (FE) static and dynamic analyses of concrete arch and gravity dams on a desktop computer and provides graphic pre- and post-processing capabilities. The FE meshes of the concrete dam, foundation rock, and the impounded water are generated automatically from a limited amount of input data. Various two- (2-) and 3-D graphics are produced to examine the accuracy of the analytical models. The results of static and dynamic analyses are displayed in graphical forms for easy interpretation and evaluation. In particular, the GDAP post-processor automatically evaluates the response-history results and extracts the critical information for presentation and further evaluation.

# 2. HARDWARE AND SOFTWARE REQUIREMENTS

GDAP software packages run on 386/486-based personal computers under MS-DOS operating systems. The specific hardware and software requirements are as follows:

### HARDWARE

- 80386-387/or 486-based Personal Computer
- 3-8 MB of Memory
- Minimum of 150 MB Hard Disk
- Color Monitor
- Mouse or Digitizing Tablet
- Tape Backup (optional)
- Laser Printer and/or Color Pen Plotter

### SOFTWARE

- MS-DOS Operating System
- AutoCAD Release 10.0 or higher

## 3. PROGRAM DESCRIPTION

The program GDAP consists of six modules as follows:

```
1. SAS2DAP    - ADSAS to GDAP Translator
2. PREPRS     - Pre-Processor Module
3. GDAP       - Analysis Program
4. INCRES     - INCompressible REServoir Program
5. POSTPRS    - Post-Processor Module
6. POSTPLT    - Post-Processor Plot Module
```

A flow chart of the program is shown in Figure 3.1. The PREPRS and SAS2DAP modules are an integral part of the GDAP program and are activated using the appropriate input options. The INCRES and POSTPRS are separate units which are linked to GDAP through input and output files. POSTPLT handles all the time-history, stress contours, and vector plots.

### SAS2DAP: ADSAS to GDAP Translator

The SAS2DAP module allows the ADSAS data files for arch dams to be used directly as an input to the GDAP program. When the input is an ADSAS data file, SAS2DAP automatically converts them into a format consistent with the GDAP input data. GDAP then uses these data to construct FE models of the dam and foundation system and to perform static analysis for the basic gravity, hydrostatic, and temperature loads. The translated data are saved on a file for further mesh refinements or additional analyses under different loading combinations.

### Pre-Processor

The pre-processor automatically generates FE meshes for the arch dam, the foundation rock, and the reservoir water from an ADSAS or GDAP input data. Alternatively, the geometry and element data may be provided manually when automatic mesh generation is not used.

2

Depending on the options selected, PREPRS will generate various 2-D and 3-D graphics that are displayed on the computer monitor with the AutoCAD software package. AutoCAD is used also for graphics editing, adding captions, cross-sectional hatching, combining various pictures, and preparing hard copy plots for the report. The following is an outline of the various graphics that can be generated by the pre-processor:

- *3-D plot of dam-foundation system*
- *3-D plot of dam model alone*
- *3-D plot of foundation alone*
- *3-D plot of half of dam-foundation system*
- *3-D plot of reservoir model*
- *3-D shrink plots of dam, foundation, and reservoir*
- *3-D view of cantilever and foundation sections*
- *Crown cantilever with line of centers*
- *Plan view of arch or horizontal sections*
- *Dam element numbers displayed on U/S and D/S faces*
- *Dam nodal numbers illustrated on both faces*

In addition, the pre-processor has an option for automatic generation of an FE mesh for the reservoir water. The reservoir model is assumed to be prismatic and extends upstream to a distance equal to at least three times the water height, and it matches the concrete nodes at the upstream face of the dam. The generated reservoir model is used as an input to the INCRES program to calculate the incompressible added mass matrix or to plot a 3-D picture of the mesh.

## GDAP Program

GDAP is an FE program specifically designed for the static and dynamic analyses of concrete dams. It includes an advanced mesh generator that automatically produces FE meshes for the dam, foundation rock, and the reservoir water from a minimum amount of input information (Figure 3.2).

The static and dynamic analyses are based on the linear-elastic material properties. The static loads include the separate or combined action of gravity, hydrostatic, temperature, silt, and the concentrated loads; and the dynamic analysis includes both the response-spectrum and time-history modal superposition methods.

3

**Dam Model**: GDAP makes two types of idealization for the automatic mesh generation of the concrete arch as shown by Ghanaat and Clough.* The first idealization assumes one element through the dam thickness and uses curved shell elements to model a thin arch dam. In the second idealization, the concrete arch is idealized by three layers of eight-node, 3-D solid elements through the dam thickness, which is more suitable for the gravity arch dams. Both element types are based on isoparametric formulation and are described by Ghanaat.**

The extended mesh generator of GDAP can handle any general three-centered arch dam of arbitrary geometry. The single- and two-centered arch dams are treated as special cases. The FE mesh of the arch is first defined on the reference surface and then is projected onto the upstream and downstream faces to get the nodal points. The reference surface is a vertical cylindrical surface passing through the upstream edge of the crest. In general, all nodes on the reference surface are arranged on horizontal sections (called mesh elevations) and on vertical planes projected from the intersection of mesh elevations with the dam-abutment interface. In addition, the program permits to add free arch and cantilevers to the mesh layout at any prescribed elevation to facilitate mesh generation of arch dams located in canyons with irregular and complex geometry. A free cantilever is defined as a typical vertical line on the reference surface that is not connected to a horizontal mesh line (mesh elevation) at its intersection with the abutment. This is specially suitable for modeling arch dams in wide canyons (Figure 3.3). Similarly, a free arch is a typical horizontal section on the reference surface that is not supported by cantilevers (or not connected to a vertical line) at its intersection with the abutment (Figure 3.4). The arches can be free at one or both ends and the free cantilevers may be added at one or both abutments (Figure 3.5).

**Foundation Model**: The effects of foundation-dam interaction are accounted for by including an appropriate portion of the foundation rock as part of the FE idealization. The GDAP mesh generator module generates a prismatic foundation mesh and permits three degrees of refinements based on the volume of rock and the number of elements used in the foundation model.

The foundation mesh is constructed on semicircular planes cut into the canyon walls in the direction normal to the dam-rock contact surface at the interface node locations (Figure 3.2). On

---

*Ghanaat, Y., and Clough, R. W. 1989. "EADAP: Enhanced Arch Dam Analysis Program", Report No. UCB/EERC-89/07, Earthquake Engineering Research Center, University of California, Berkeley, CA.
**Ghanaat, Y. 1993. "Theoretical Manual for Analysis of Arch Dams" (Technical Report ITL-93-1), U. S. Army Engineer Waterways Experiment Station, Vicksburg, MS.

4

each of these planes, a semicircle is drawn from the dam-rock interface with a radius equal to the dam height for the foundation mesh types one and two (Figure 3.6), and equal to two times of dam height for the mesh type three (Figure 3.7). Three-dimensional, eight-node brick elements are used for all three foundation mesh types. The number of foundation elements on each semicircular plane for the three mesh types are 8, 13, and 18, respectively; the number of elements on each plane are increased to 10, 15, and 20 when eight-node brick elements are also used to model the concrete arch.

Furthermore, there are three options available for the orientation of the semicircular planes at the crest level. These planes may have downward slopes (i.e. stay normal to the dam-rock contact surface) or may be rotated upward to a horizontal or upward sloping position to represent the actual canyon shapes (Figure 3.8).

## INCRES Program

INCRES is an FE program for calculating an equivalent added mass matrix for the incompressible reservoir water. The added mass of water is obtained from the hydrodynamic pressure distributions on the face of the dam by solving the pressure wave equation. The calculated added-mass matrix is then supplied as additional input data to GDAP to account for the dam-water interaction in the dynamic analysis.

**Reservoir Water Model:** The GDAP pre-processor automatically generates a prismatic reservoir FE model for calculating the added mass of the impounded water. The reservoir water model consists of liquid elements arranged in three or more successive layers with liquid nodes located to correspond with concrete nodes on the dam-reservoir interface. When the mesh generation capability of GDAP is used, the reservoir water mesh extends upstream to a distance equal to the number of fluid layers times the water depth. Figure 3.9 shows two prismatic reservoir models for a three-layer and a five-layer liquid mesh. The dam-water interface is modeled by 8-node, 2-D curvilinear elements, whereas the body of water is represented by 16-node, 3-D liquid elements with nodal pressures being the unknowns. Both element types are based on the isoparametric formulation and are described by Ghanaat.* Topographic features of the reservoir may be included in the FE model by manually modifying the generated nodal coordinates to match the actual shape of the canyon.

---

* Op. cit. p.4.

## Post-Processor

The GDAP post-processor (POSTPRS) converts the results of static and dynamic analyses into appropriate plots and contours for easy review and evaluation.

POSTPRS reads the static deflections and vibration mode shapes calculated by GDAP and generates the associated AutoCAD plot files. For the element stresses, however, the post-processing is carried out in two steps. In the first step, POSTPRS reads the element stresses and separates them into plot files including the upstream and downstream arch, cantilever, shear, and principal stress components. In the second step, the stress plot files are used as input to the POSTPLT plotting routine to generate stress contour plots of the arch and cantilever stresses, vector plots of the principal stresses, or the time-history plots of the nodal displacements and the element stresses.

Furthermore, the post-processor includes evaluation criteria for processing the large amount of data produced in a typical response-history analysis. It automatically retrieves the envelope of the maximum and minimum stress values that could occur at any instant of time, identifies all significant concurrent stresses, recovers stress histories at all critical locations and at their corresponding points on the opposite face of the dam, provides statistics regarding the number of stress cycles exceeding the allowable stress (specified by the user), and calculates the excursion time of stress cycles beyond the allowable values. A list of available features are summarized :

- *Plot of nodal displacements and mode shapes.*

- *Contour plots of the arch and cantilever stresses due to the separate and combined action of various static loads.*

- *Contour plots of the envelope arch and cantilever stresses due to the response spectrum dynamic analysis only and the response spectrum plus static loads.*

- *Contour plots of the envelope arch and cantilever stresses due to the dynamic (response-history) only and the dynamic plus static loads.*

- *Contour plots of concurrent stresses at critical instants of time due to the dynamic-only and the dynamic-plus static loads.*

- *Vector plots of static, dynamic (response-history), and static-plus dynamic principal stresses.*

- *Time-history plots of the input earthquake motions and the critical nodal displacements and element stresses.*

6

- *Statistics on number of stress cycles exceeding allowable stress and the corresponding excursions of these stress cycles beyond specified limits.*

## POSTPLT

POSTPLT is an interactive plotting routine which runs under MS-DOS. The program prompts for the input data and the names of the files containing the desired response quantities. The response quantities include static or dynamic stress files or the nodal displacement and element stress histories as generated by the post-processor. Stresses due to several load cases may be specified as input. In that case, such stresses are combined and the desired plots are produced for the combined action.
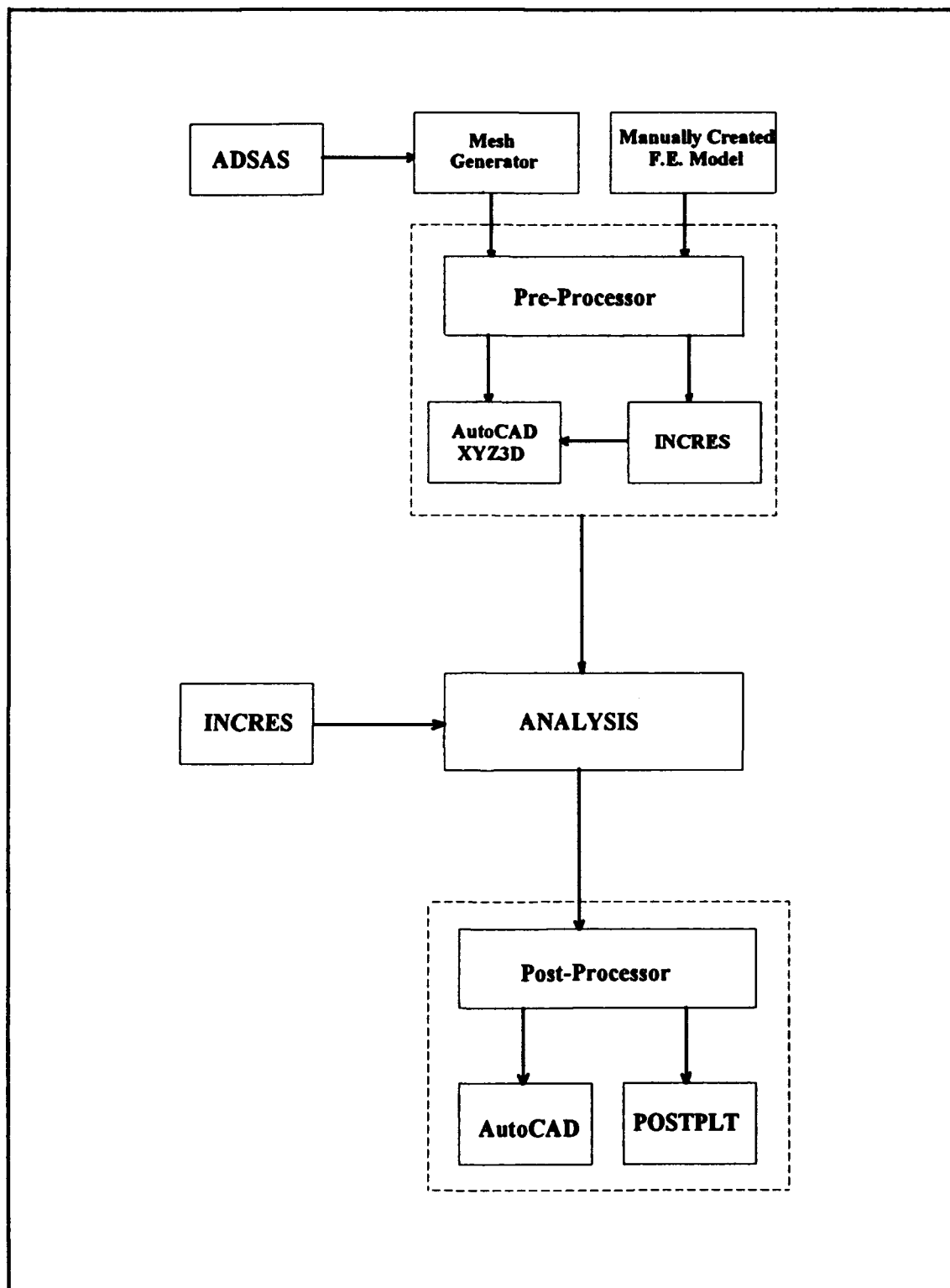
Figure 3.1  Flow Chart of GDAP Program

8
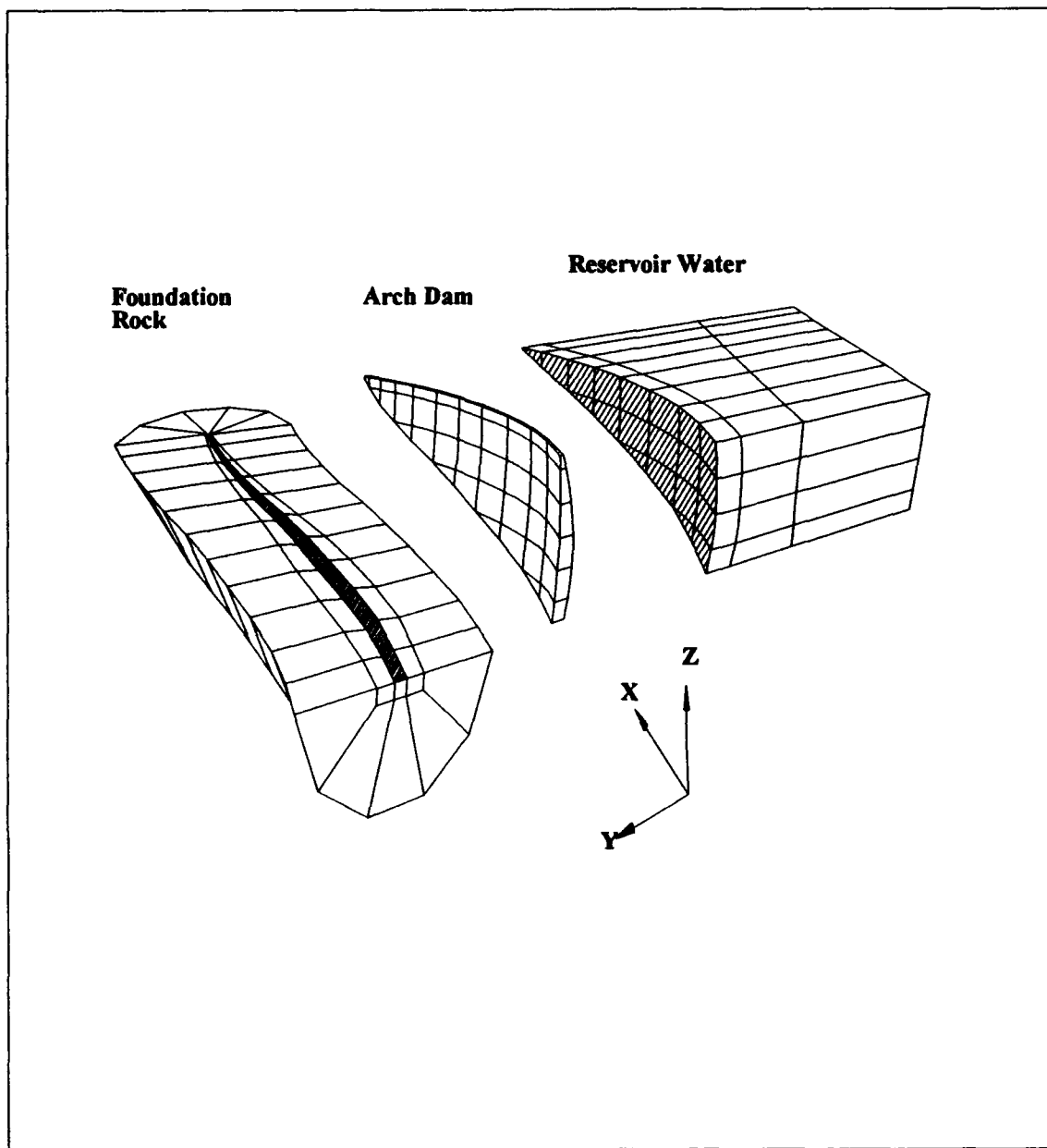
Figure 3.2  Finite Element Models of Half of Dam, Foundation, and Fluid Domain

9

(a) Without Free Cantilevers

Free Cantilever
(no arch at this elevation)

(b) With Free Cantilevers
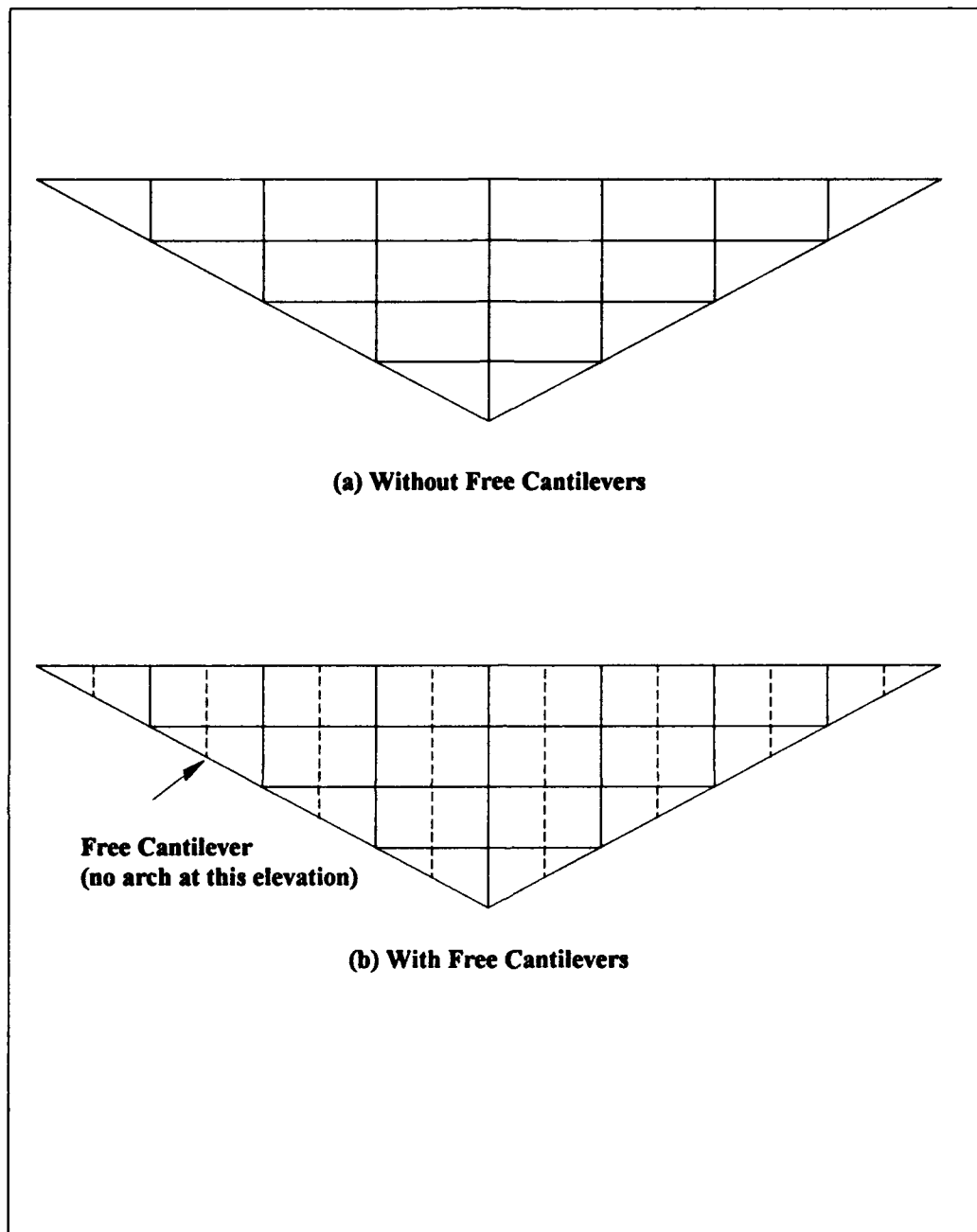
Figure 3.3  Finite Element Mesh of an Arch Dam in Wide Canyon

**Free Arch
(no cantilever at this elevation)**

**(a) Without Free Arches**　　　　　**(b) With Free Arches**
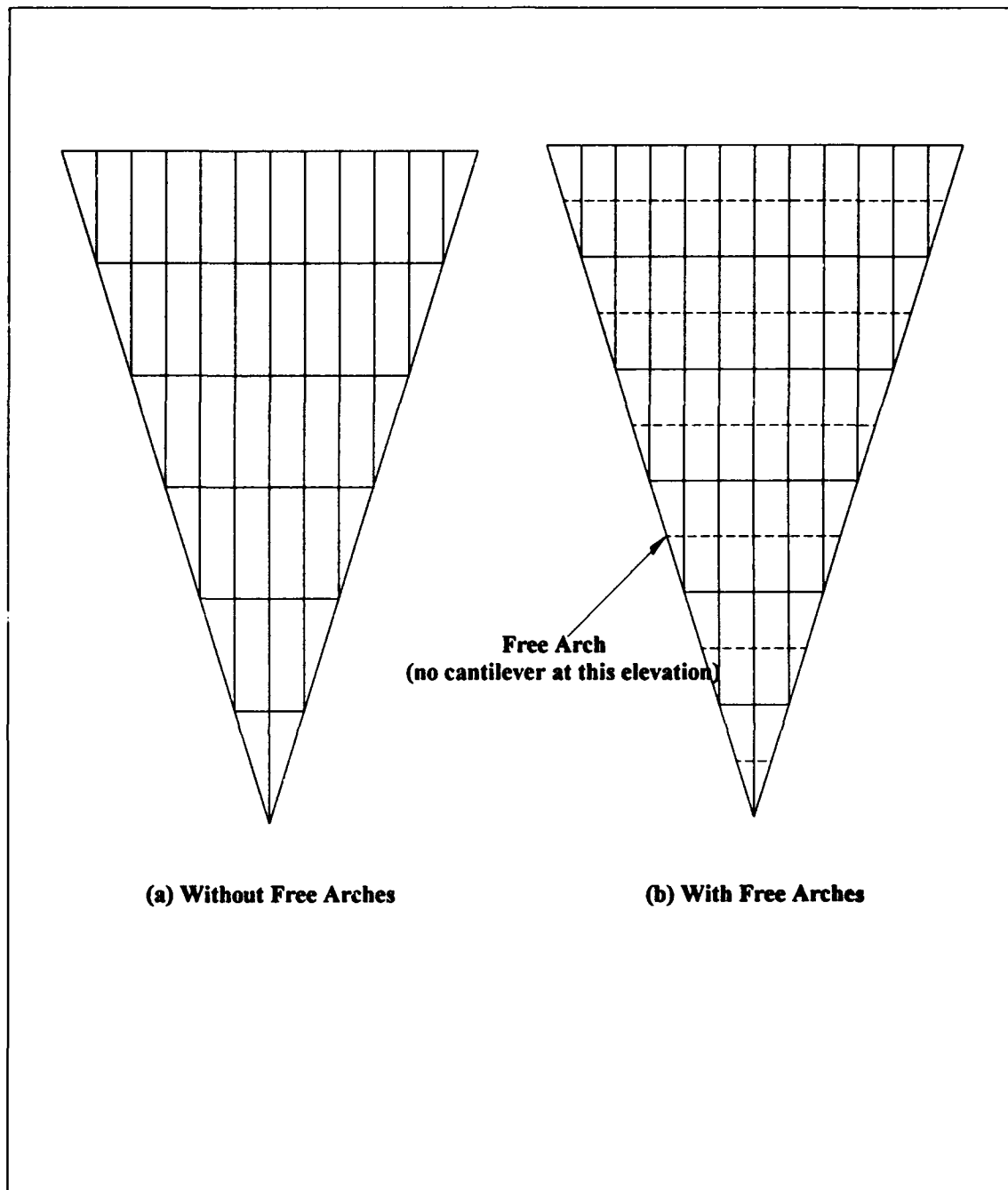
Figure 3.4　Finite Element of an Arch Dam in Narrow Canyon

**(a) Without Free Arches and Cantilevers**

**Free Arch**

**Free Cantilever**

**(b) With Free Arches and Cantilevers**

**Free Arch on One Side**
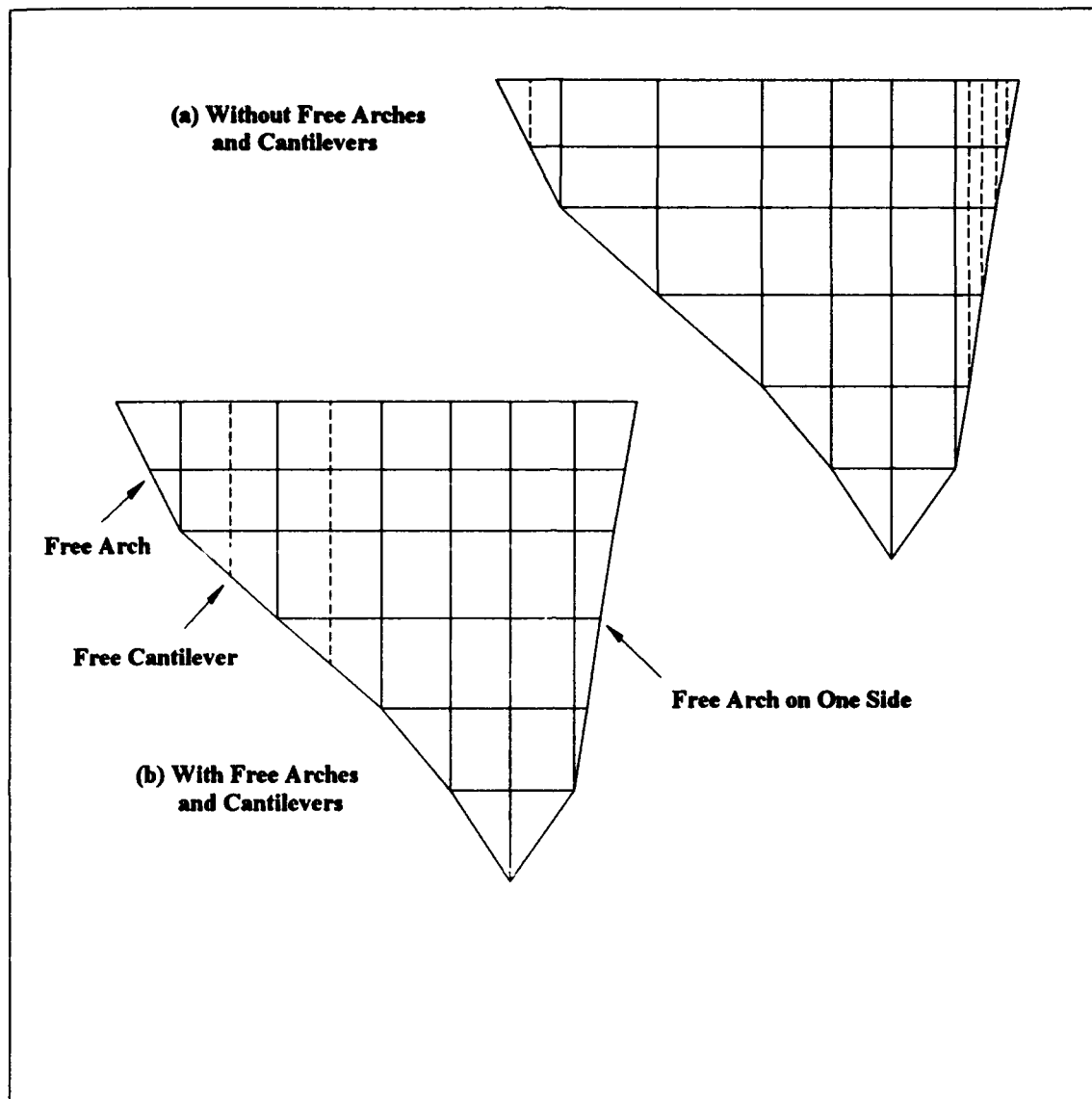
Figure 3.5  Finite Element of an Arch Dam in Irregular Canyon

12
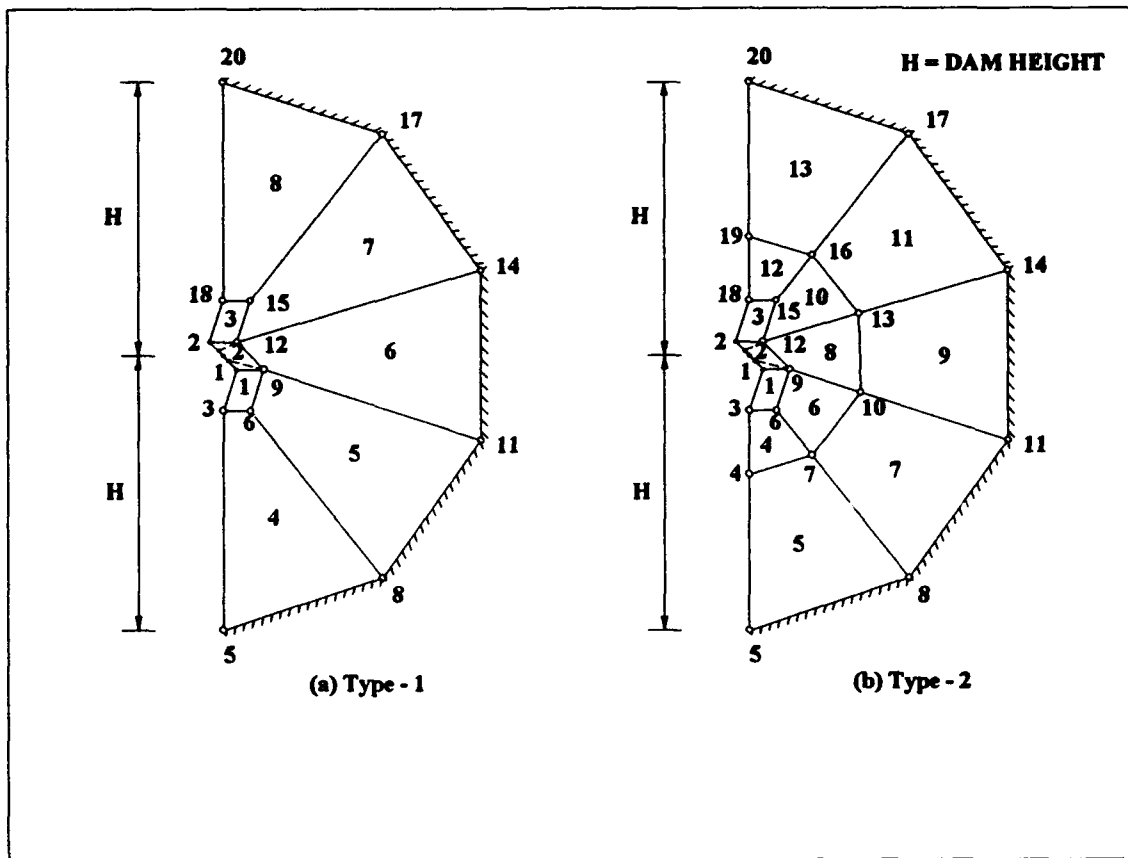
Figure 3.6  GDAP Foundation Mesh Types 1 and 2 on Sections Normal to Dam-Rock Interface
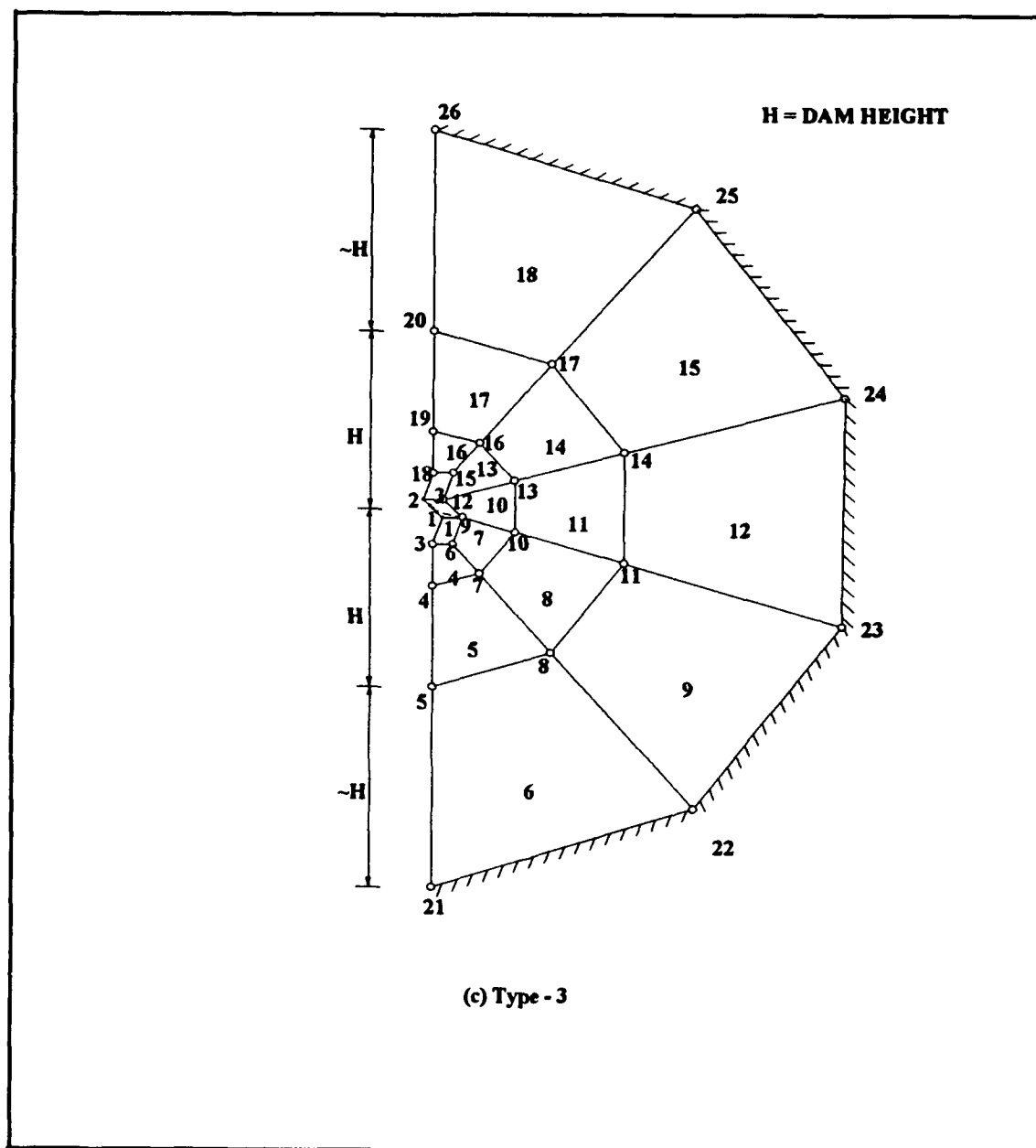
Figure 3.7  GDAP Foundation Mesh Type 3 on Section Normal to Dam-Rock Interface

14

(a) Downward Sloping Foundation

(b) Horizontal (level) Foundation

(c) Upward Sloping Foundation
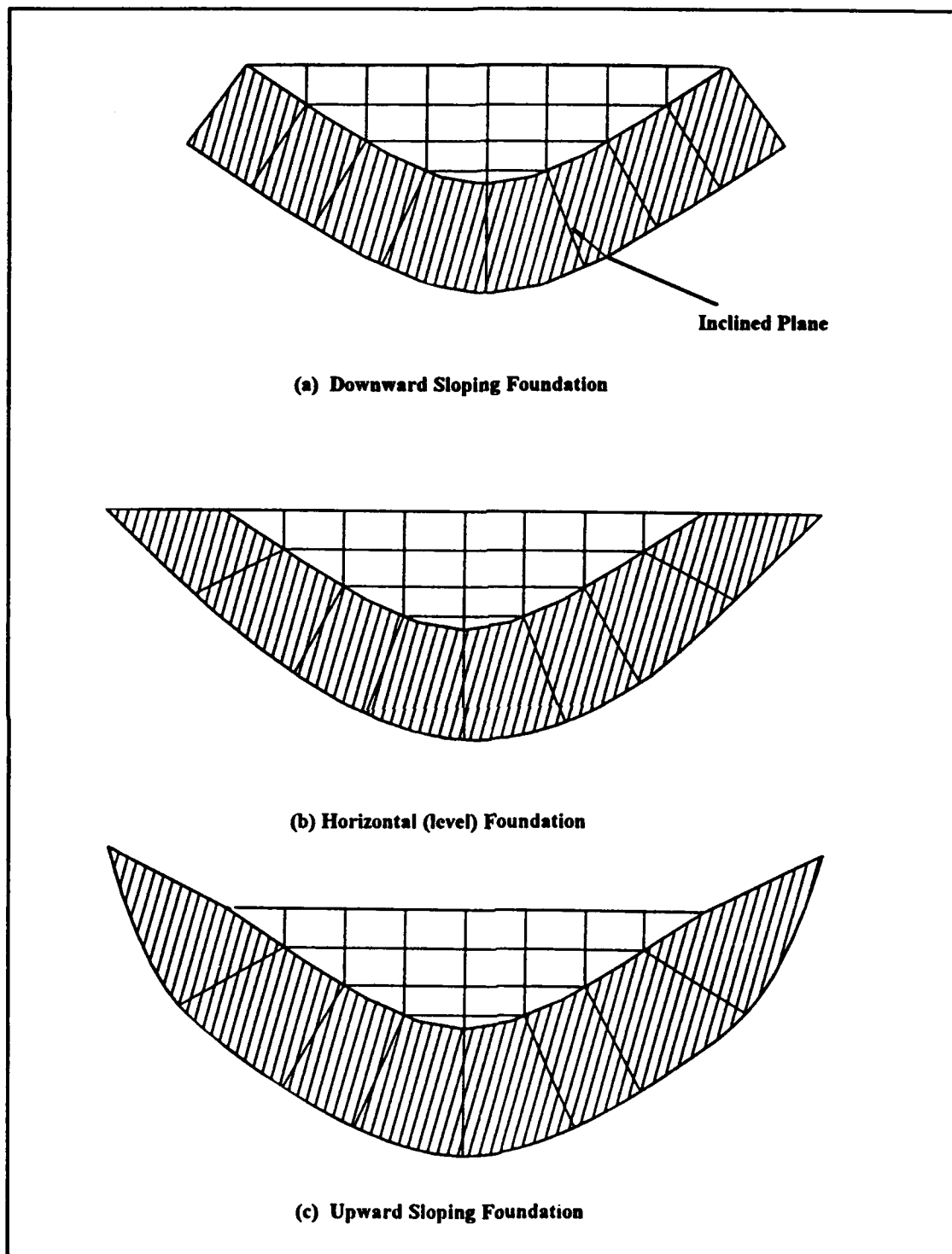
Figure 3.8   GDAP Foundation Rock Models

(a) Three Fluid-layer Model

(b) Five Fluid-layer Model
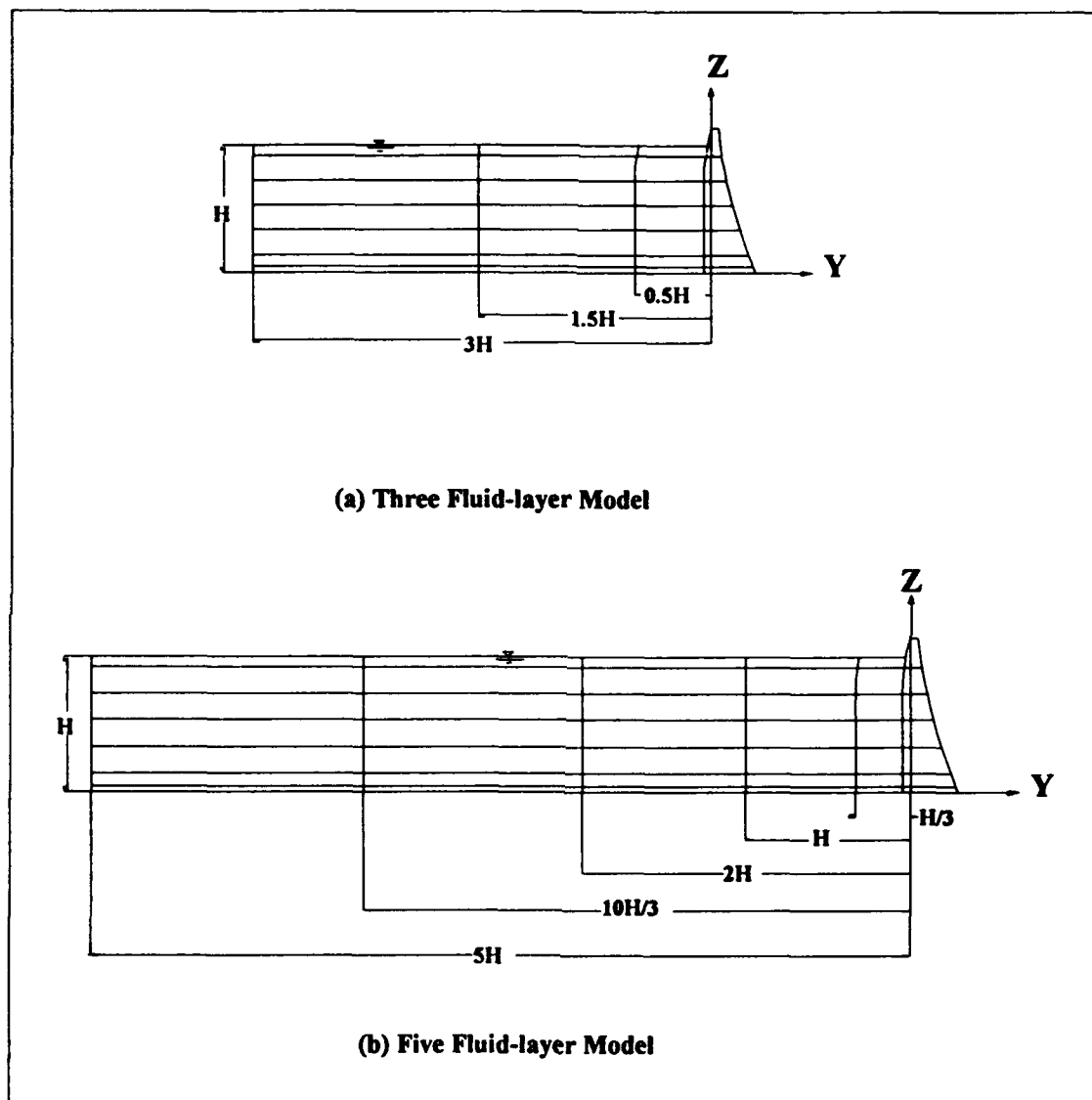
Figure 3.9   Section View of Finite Element Reservoir Water Model

# 4. OUTLINE OF STATIC ANALYSIS PROCEDURE

The GDAP program performs a linear-elastic static analysis of any arbitrary arch dam-foundation system. The dam-foundation system is idealized as an assemblage of FE's as described in Section 3. The geometric and element data are specified automatically by the mesh generation capabilities or manually in accordance with the format description given in Section 6. Linear-elastic material properties are assumed for both the concrete and the foundation rock. Shell elements in the concrete arch are assumed to be isotropic, but orthotropic properties may be assumed for the 8-node solid elements representing the foundation rock. A detailed description of the static analysis and the static loads is given by Ghanaat.[*]

The following static loads are considered and may be applied separately or in any arbitrary combination:

- *Gravity load*
- *Water load*
- *Temperature Load*
- *Silt Load*
- *Concentrated Load*
- *Ice load*

The results of static analysis include nodal displacements, element stresses, and the reaction forces or thrusts at the dam-foundation interface due to the applied loads. For each thick-shell nodal point, five displacement components corresponding to three translations and two rotations are provided, whereas for all other nodal points only three translations are given. For each element, the stresses are given at several points referred to as stress points. [Refer to Figures 6.9 and 6.10.] All element stresses are calculated with respect to a set of local axes, except at the center of the 8-node solid elements where they are given in reference to the global axes. The local stresses are calculated on the upstream and downstream faces of the dam and include the arch, cantilever, normal, shear, and the principal stresses. The reaction forces can be obtained at any nodal point, but they are normally needed at the dam-foundation interface for the stability analysis of the abutment rock mass.

---

[*] Op. cit. p.4.

# 5. OUTLINE OF DYNAMIC ANALYSIS PROCEDURE

The GDAP program performs linear-elastic dynamic analysis for concrete arch, gravity arch, and multiple arch dams. It accounts for the significant dynamic interaction effects of the foundation rock and the reservoir water. The dynamic response is obtained by solving the following system equations of motion:

$$(M + M_a)\ddot{U} + C\dot{U} + KU = -(M + M_a)r\ddot{U}_g$$

where

      M = Mass matrix of the concrete arch system

      $M_a$ = Added mass of the reservoir water

      C = Viscous damping matrix

      K = Stiffness matrix of dam-foundation system

      U = Nodal displacement vector

      $\dot{U}$ = Nodal velocity vector

      $\ddot{U}$ = Nodal acceleration vector

      $\ddot{U}_g$ = Vector of earthquake accelerations

      r = Influence coefficient matrix

The foundation rock is assumed massless, and thus only its stiffness is included in the equations of motion. The added mass of the incompressible water may be calculated by the generalized Westergaard method or the FE formulation. The Westergaard added mass is calculated by the GDAP program, whereas the FE added-mass matrix is computed by the INCRES program as described in Section 3.

The solution of the system equations of motion is based on the mode-superposition method of dynamic analysis. Two types of mode-superposition methods are provided in the program: the response-spectrum and the response-history methods. In both methods, first the equations of motion are transformed to the uncoupled modal coordinate forms using the free vibration mode shapes of the dam-water-foundation system. Then the response for each uncoupled equation is computed and these modal responses are superimposed to obtain the total response of the structure. In the response-spectrum method, only the maximum response of the structure is calculated, which is obtained by combining the modal maxima computed for each mode of vibration by the square-root-of-the-sum-of-the-squares (SRSS). In the response-history analysis, however, the complete response history of each mode for the duration of the input ground acceleration is calculated by the

linear acceleration step-by-step integration method. The resulting modal displacements and stresses at each time-step are then superimposed to obtain the total response history of the structure.

The earthquake response spectrum and the acceleration time-histories are used as seismic input for the response spectrum and the response-history analyses, respectively. Any single component of the selected seismic input or all three components may be applied in the dynamic analysis. The results of analysis include the envelope nodal displacements and element stresses for the response spectrum method and the time-histories of nodal displacements and element stresses for the response-history analysis. In response-history analysis, the output also includes the maximum and minimum response values and the associated time-steps.

# 6. GDAP INPUT DATA DESCRIPTION

The input data for the GDAP program and its pre-processor are prepared according to the free-format specification given in this section. Each record contains one or several input values that are separated by a comma or one or more contiguous blanks. The blanks are treated as separators, thus the null values should be specified as zeros and not as blanks. The I/O system formats each input value using the data type and the field width of the corresponding input list item.

## GDAP INPUT DATA

## A. TITLE RECORD

TITLE        Information to be printed as the output header. Title is limited to 72 characters.

## B. MASTER CONTROL PARAMETERS

One or both of the following records are required depending on the type of the input data.

### Record B.1 - Model Definition and Analysis Type

This record is always required.

NUMNP     Total number of nodal points in the dam-foundation model. Enter zero if Mesh Generation is used (Note-a).

MTOT       Size of the available blank **COMMON** block (Note-b).

NELTYP    Number of different element types to be used (Note-c).

LL          Number of static load cases; enter zero in dynamic analysis.

NF          Number of undamped natural frequencies to be calculated or number of modes to be considered in the time-history or response-spectrum analysis. Enter zero for static analysis.

NDYN      Analysis type selection:

| Static Analysis | Eigen Solution | Time History | Response Spectrum | Graphics Pre-processing |
|---|---|---|---|---|
| 0 | 1 | 2 | 3 | 4 |

NLM      Number of mesh elevations defining the dam model. Enter zero if mesh generation is not used.

NLU       Mesh elevation number at the base elevation of a U-shaped valley (Figure 6.1). Enter zero for V-shaped valley.

**NEQEST**    Estimated number of degrees of freedom (DOF's). Enter zero if no estimate is available. The execution halts if the estimated and computed DOF's do not match (Note-d).

**IMODE**    Restart option for dynamic analysis:
= 1, Mode shapes and frequencies are stored or read from the restart TAPE10.DAT.
= 0, otherwise and for static analysis.

**IPRM**    Option for mode-shape printout in dynamic analysis:
= 0, mode shapes are printed; = 1, otherwise.

**ESTVOL**    Estimated total volume of all elements. Enter zero if no estimate of element volumes is available. The execution halts if estimated and computed volumes do not match with a 10E-4 accuracy (Note-e).

**MESH**    Dam mesh type:

= 0, manual data input, mesh generation is not used;
= 1, dam is modeled by the combination of 16-node shell and thick-shell elements, one element through the dam thickness is used;
= 3, dam is modeled by 8-node brick elements, three elements through the dam thickness are used.

**MESHFN**    Foundation mesh type (Figures 3.6 and 3.7):

| Rigid | Type-1 | Type-2 | Type-3 |
|-------|--------|--------|--------|
| 0     | 1      | 2      | 3      |

Enter zero when mesh generation is not used.

**IADMAS**    Added-mass type selection for dynamic analysis:

| Empty Reservoir or Static Analysis | Generalized Westergaard Added mass | FE Added mass |
|-----------------------------------|-----------------------------------|---------------|
| 0                                 | 1                                 | 2             |

For the FE option, a separate reservoir model should first be developed. The added mass is then obtained by running the INCRES program and is stored on TAPE12.DAT as an input for the GDAP dynamic analysis. When the Westergaard option is selected, the added mass is calculated by the GDAP program and no separate reservoir model is needed.

**WATL**    Z-coordinate of the water level.

**WDEN**    Water weight density.

21

## Notes:

a)   For THKSHEL elements only, the surface nodes are counted. When mesh generation is used, NUMNP is automatically calculated by the program and consists of the nodal points for the dam with foundation mesh type-3.

b)   MTOT controls the number of blocks and number of equations in each block for the out-of-core solution. Smaller MTOT values result in larger number of blocks with smaller number of equations per block. Depending on the hardware resources, it could be set to any number in the range of 10,000-200,000.

c)   Maximum of three element types can be specified: 8-node brick, element type-1; 16-node shell, element type-2; thick-shell, element type-3.

d)   NEQEST may be used for checking the generated data. If set to a nonzero value other than the actual number of DOF's, nodal coordinates, ID array, and the element data are generated and then the execution stops.

e)   ESTVOL may be used for further examination of the generated element data to identify any excessive element distortions. If set to a nonzero value other than the actual total volume of all elements, stiffness and mass matrices for each element are calculated and the elements volumes and connectivities are printed out. Then the execution stops and no response is calculated.
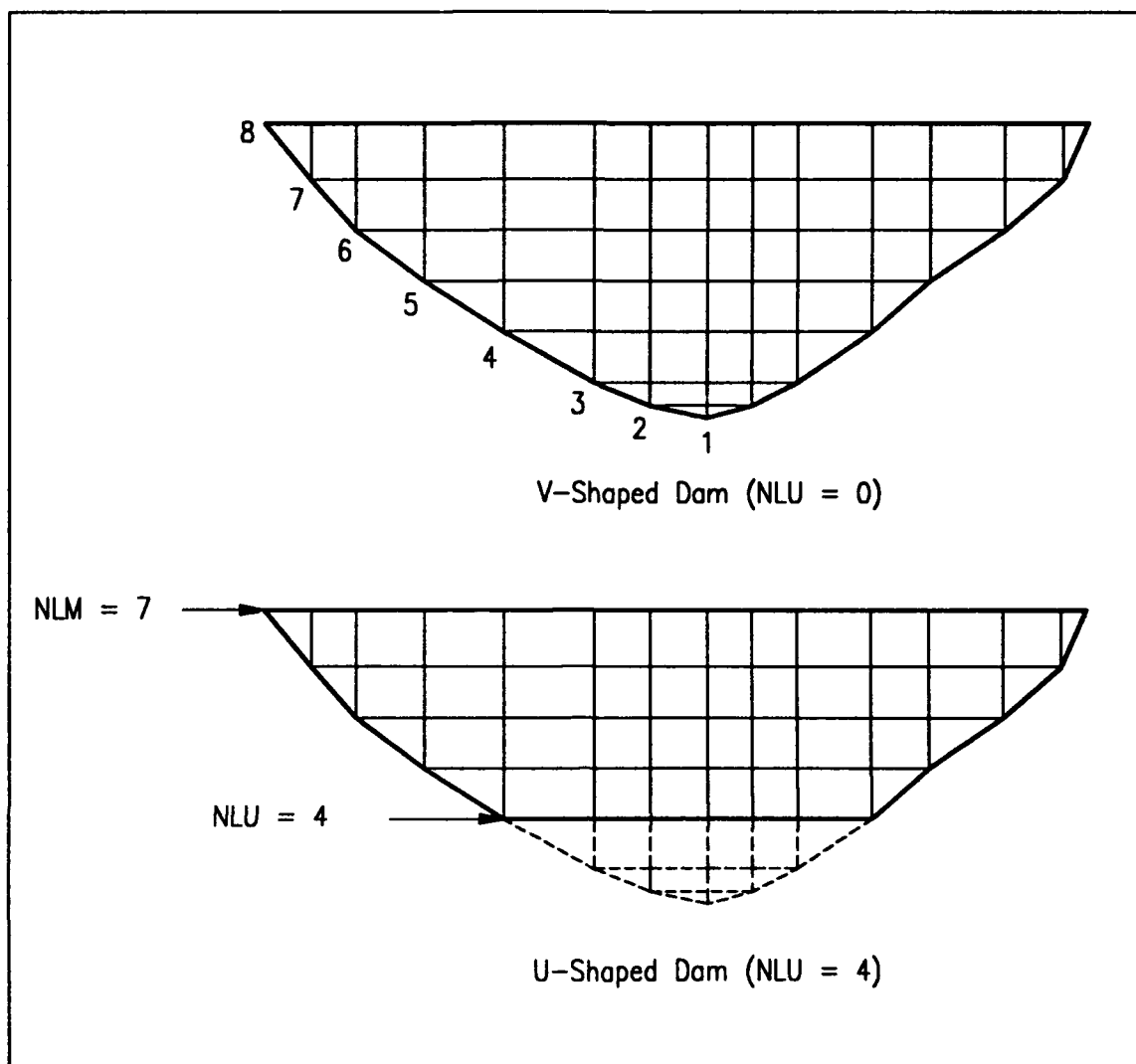
Figure 6.1  V-Shaped, U-Shaped Dams

Note:  Dashed lines are fictitious grids used to construct FE mesh for the dam in a U-shaped valley.  The base of the dam is located at elevation NLU.

## Record B.2 - Dynamic Analysis with Restart Option

This record is required for dynamic response calculation for which mode shapes, frequencies, and element data are read from the restart files. The first five parameters shown specify the problem size and are retrieved from the output of the free vibration analysis. NF1 and NF2 are the mode selection parameters. They allow inclusion of a single mode, a range of modes, or all the calculated modes in the dynamic response calculation.

MBAND       1/2 bandwidth of the system of equilibrium equations.

NUMEL       Total number of elements (dam plus foundation).

NEQ         Number of equations or DOF's.

N3DDAM      Number of 3-D brick elements in dam.

N3DFN       Number of 3-D brick elements in foundation.

NSHEL2      Number of 3-D shell elements.

NSHEL3      Number of thick-shell elements.

NF1         Starting mode number at which response calculation will begin,
            ( $1 \leq NF1 \leq NF$ ).

NF2         Ending mode number at which response calculation will stop,
            ( $NF1 \leq NF2 \leq NF$ ).

## C. MESH GENERATION INPUT DESCRIPTION

Skip this section if mesh generation is not used or if this is a dynamic response calculation for which the structural data and frequencies and mode shapes are read from the restart files.

### Record C.1 - Reference Surface Data

| | |
|---|---|
| **RI** | Radius of the inner portion of the reference surface* (Figure 6.2). |
| **RO(1)** | Radius of the right outer portion of the reference surface. |
| **RO(2)** | Radius of the left outer portion of the reference surface. |
| **NL** | Number of design elevations. |
| **IEL** | = 1, same compounding angles are specified at all elevations. |
| | = 0, compounding angles differ for each elevation. |
| **IRL** | = 1, same compounding angles are specified for the right and left portions of the dam. |
| | = 0, otherwise. |
| **IIE** | = 1, same compounding angles are specified for intrados and extrados faces of the dam. |
| | = 0, otherwise. |
| **NRL** | = 1, same radius is specified for the right and left portions of intrados and extrados. |
| | = 0, otherwise. |
| **KFN** | Flag for orientation foundation planes at the top of dam: |
| | = -1, standard downward inclined plane (Figure 3.8); |
| | = 0, horizontal plane at the crest elevation; |
| | = 1, upward inclined plane. |
| **ISYM** | = 0, nonsymmetric dam, or when symmetry is not used. |
| | = 1, symmetric dam modeled as a symmetric structure with symmetric boundary conditions (BC's) along the crown section. |
| | = -1, symmetric dam modeled as a symmetric structure with antisymmetric BC's along the crown section. |

---

* Reference surface is a vertical cylindrical surface which passes through upstream edge of the crest.

25

## Record C.2 - Compounding Angles and Angles to Abutments

One record is required for each design elevation to specify the compounding angles and angles to abutments (Figure 6.2). The sequence of records corresponds to increasing order of elevations.

| | |
|---|---|
| **EL(I)** | Elevation i. |
| **FCI(I,1)** | Compounding angle of the right-intrados arc at elevation i, $\phi_1$. |
| **FCI(I,2)** | Compounding angle of the left-intrados arc at elevation i, $\phi_2$. |
| **FCE(I,1)** | Compounding angle of the right-extrados arc at elevation i, $\phi_3$. |
| **FCE(I,2)** | Compounding angle of the left-extrados arc at elevation i, $\phi_4$. |
| **FA(I,1)** | Angle to the right abutment at elevation i, $\phi_5$. |
| **FA(I,2)** | Angle to the left abutment at elevation i, $\phi_6$. |

## Notes:

1. If IEL = 1 (Record C.1), compounding angles for the first elevation only is required.

2. If IRL = 1 (Record C.1), compounding angles for the right arcs only is required.

3. If IIE = 1 (Record C.1), compounding angles of the intrados only is required.

## Record C.3 - Temperature Data

Two sets of data records are required to specify the temperature data at the design elevations.

The first data set corresponds to the upstream face and as many records as required are supplied to specify the temperature values at all design elevations. Temperature values are specified in the sequence of increasing elevations.

The second set of records corresponds to the downstream face. Temperature values are specified in exactly the same way as described above.

Zero values should be provided, if temperature variation is not considered in the analysis.
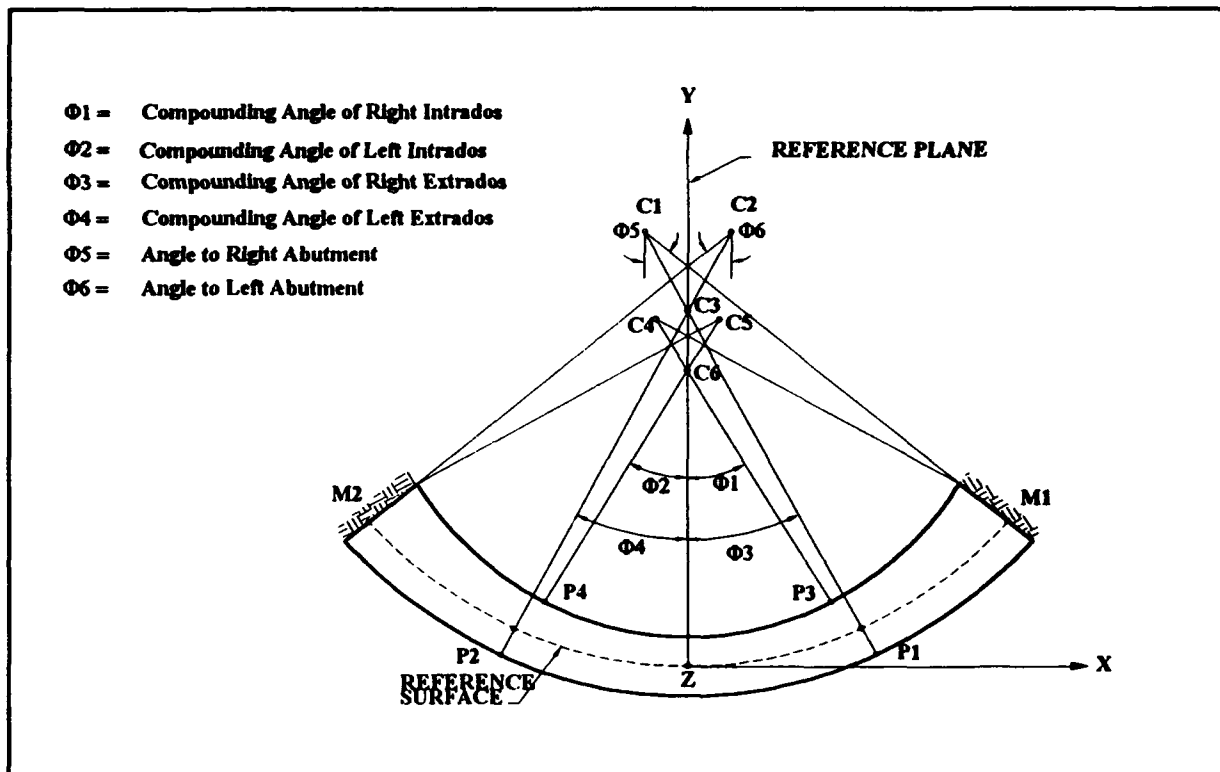
Figure 6.2  Typical Horizontal Section of a Three-Centered Arch Dam

## Record C.4 - Mesh Elevations

Mesh elevations are specified in increasing sequences, and as many records as required should be supplied. A maximum of 20 mesh elevations may be specified. The following data are supplied for each mesh elevation at which there is a supported or unsupported arch or cantilever line as shown in Figure 3.5.

**ICNTRL(I,1)**    = 0,  no arch line at mesh elevation i.
                           = 1,  an arch line is placed at elevation i.

**ICNTRL(I,2)**    = 0,  no cantilever line on the right side at this elevation.
                           = 1,  a cantilever line is placed on the right side at this elevation.

**ICNTRL(I,3)**    = 0,  no cantilever line on the left side at this elevation.
                           = 1,  a cantilever line is placed on the left side at this elevation.

**ELM(I)**           Mesh elevation i.


## Record C.5 - Intrados and Extrados Arcs

One record is required for each design elevation to specify the radius and Y-coordinate of the center of each arc (Figure 6.2). The sequence is according to the increasing order of the elevations.

**YII**              Y-coordinate of center of intrados inner arc (point $C_6$ in Figure 6.2).

**YEI**              Y-coordinate of center of extrados inner arc (point $C_3$).

**RII**              Radius of intrados,  inner arc.

**REI**              Radius of extrados,  inner arc.

**RIO(1)**           Radius of intrados,  right outer arc.

**REO(1)**           Radius of extrados,  right outer arc.

**RIO(2)**           Radius of intrados,  left outer arc.

**REO(2)**           Radius of extrados,  left outer arc.

**Notes:**

If NRL = 1 (Record C.1), radii of the left outer arc for intrados and extrados may be set to zero.

28

## Record C.6 - X-Coordinate of Center of Inner Arcs

Two set of records are required to specify the X-coordinate of the center of inner arcs at the design elevations.

The first set corresponds to the intrados inner arc. Coordinate values are specified in the sequence of increasing elevations and as many records as required should be supplied.

The second set of records specify the X-coordinates of the center of the extrados inner arc. Same procedures mentioned above apply to this set.


## Record C.7 - Material Properties of Elements

The following set of records specifies the material property identification numbers for each element type.


### Record C.7.1 - Eight-node Brick Elements of Dam

For mesh type-1 (MESH=1, Record B.1), no record is required.

For mesh type-3 (MESH=3), when all eight-node brick elements of the dam have the same material properties (i.e. homogeneous concrete), two zeros, one for NLL and the other for MATT, as described, should be supplied. In this case, material number 1 will be assigned to all eight-node brick elements of dam.

For mesh type-3, when eight-node brick elements of the dam have different material properties, one record should be assigned to each group of elements having the same material properties according to the following format:


**NLL**          Element number.

**MATT**         Material identification number.


## Note:

The sequence of records should correspond with increasing order of the element numbers. If a group of successive elements have the same material numbers, only material record for the first element in the group is needed. The sequence of records should be terminated by two zeros, unless the material number for the last element is supplied.

## Record C.7.2 - Eight-node Brick Elements of Foundation

For the case with rigid foundation (MESHFN=0), no record is needed.

For MESHFN > 0 and both concrete arch dam and foundation rock are assumed to be homogeneous, two zeros should be supplied for NLL and MATT as described. In this case, material number 1 (if MESH = 1) or 2 (if MESH = 3) is assigned to eight-node brick elements of the foundation.

For MESHFN > 0 and either concrete arch dam or foundation rock is not homogeneous, a set of records should be supplied to specify the material numbers of different foundation elements. These records follow the same format described in Record C.7.1.

## Record C.7.3 - 3-D Shell Elements

For the MESH not equal to 1, no record is required.

For MESH = 1 and all 3-D shell elements having the same material properties, NLL and MATT are set to zero; and the material number 1 is assumed for all 3-D shell elements.

For MESH = 1 and 3-D shell elements having different material properties, one record is supplied for each group of elements having identical material properties. These data are prepared according to the format described for Record C.7.1.

## Record C.7.4 - Thick- Shell Elements

Follow the procedure presented for the 3-D shell elements.

# D. MANUAL PREPARATION OF NODAL COORDINATES AND TEMPERATURE DATA

Skip this section if mesh generation is used. Otherwise, one record per node is required unless some nodes are to be generated.


## Record D.1 - Nodal Coordinates and Temperature Values

**NODE**             Node number.

**COORD(NODE,1)**    X-coordinate of NODE.

**COORD(NODE,2)**    Y-coordinate of NODE.

**COORD(NODE,3)**    Z-coordinate of NODE.

**COORD(NODE,4)**    Temperature value of NODE.


These records are supplied in increasing node number sequence. However, if a group of records is omitted, the coordinates of the corresponding nodes are generated at equal intervals on a straight line connecting two nodes for which coordinates have been supplied.


## Record D.2 - Boundary Conditions and Adjacent Node Data

One record per node is supplied, unless for some nodes the adjacent nodes and boundary conditions are to be generated.

**NODE**       Node number.

**NADJ**       Adjacent node number (see the following notes).

**ID(NODE,1)**    X-translation fixity code:
                    = 0, free;
                    = 1, fixed.

**ID(NODE,2)**    Y-translation fixity code.

**ID(NODE,3)**    Z-translation fixity code.

**ID(NODE,4)**    Local x-rotation fixity code.

**ID(NODE,5)**    Local z-rotation fixity code.

## Notes:

For each nodal point in the system an adjacent node number (NADJ) is defined as follows:

1. If it is a primary node of a thick-shell element in which 5 DOF's of the corresponding midsurface node are retained (no connection to a 3-D element at that nodal point), NADJ will be the global nodal point of the corresponding adjacent node.

2. If it is a primary node of a thick-shell element which is connected to a 3-D element at that nodal point, NADJ will be equal to the global nodal point of the corresponding adjacent node with negative sign.

3. For all nodal points other than those just mentioned , NADJ will be zero.

If NADJ ≤ 0, rotation DOF's ( ID(NODE 4) and ID(NODE 5) ) are set to zero.

These records are supplied in increasing node number sequence. However, if a group of records is omitted between a pair of nonconsecutive nodes, the missing information is generated by the program as follows:

1. The boundary conditions will be the same as those on the first record of the pair.

2. The adjacent node numbers will be generated by linear interpolation between adjacent node numbers on the given pair of records.

## E. MODIFICATION OF NODAL POINT DATA

The previously generated nodal coordinates, temperature values, and the fixity data may be modified by supplying the following information.

### Record E.1 - Control Data

**MODC**      Number of nodes for which the coordinates are to be modified.

**MODB**      Number of nodes for which fixity and adjacent node numbers are to be modified.

**MODT**      Number of nodes for which temperature values are to be modified.

**IPPR**      Code for printout of nodal data:
= 0, nodal coordinates, fixity, and temperature values are printed;
= 1, no printout.

Supply zero for each parameter for which no modification is required.

### Record E.2 - Coordinate Modification

A total of MODC records is required. These data override previously generated or read in nodal coordinates. Each record corresponds to one nodal point. Arbitrary sequence may be used.

**NODE**      Node number.

**COORD(NODE,1)**      X-coordinate.

**COORD(NODE,2)**      Y-coordinate.

**COORD(NODE,3)**      Z-coordinate.

### Record E.3 - Temperature Modification

A total of MODT records is required. These data override previously generated or read in temperature values. Each record corresponds to one nodal point. Arbitrary sequence may be used.

**NODE**      Node number.

**COORD(NODE,4)**      Temperature value.

## Record E.4 - Modification of Fixity and Adjacent Nodes

A total of MODB records are used to override previously generated boundary conditions and adjacent node data. One record is needed for each node; arbitrary node sequence may be used.

| | |
|---|---|
| **NODE** | Node number. |
| **NADJ** | Adjacent node number. |
| **ID(NODE,1)** | X-translation fixity code:<br>= 0, free;<br>= 1, fixed. |
| **ID(NODE,2)** | Y-translation fixity code. |
| **ID(NODE,3)** | Z-translation fixity code. |
| **ID(NODE,4)** | Local x-rotation fixity code. |
| **ID(NODE,5)** | Local z-rotation fixity code. |

If NADJ $\leq$ 0, rotation DOF's should be set to zero.

## F. THICKNESS CHANGE

The program can handle a condition where thick-shell elements of different thicknesses are connected, as shown in the drawing at the bottom of this page. In this case, the total nodes in the structure include all surface nodes of thick-shell elements. The midsurface nodes of each pair of elements along the thickness change are assumed to coincide at points marked by $x$ in the drawing. In the input data, fixity condition and concentrated loads associated with primary nodes of one of the elements, say $i$, $j$, and $k$, will refer to those of the midsurface. The primary nodes of the other elements, $l$, $m$, and $n$, will be fixed.

One record is required for each midsurface node along the thickness change. Fixed and free nodal points are selected so that $J > I$. This set of records must be terminated by a record with zeros for I and J.

I               Corresponding fixed primary node.

J               Corresponding free primary node.
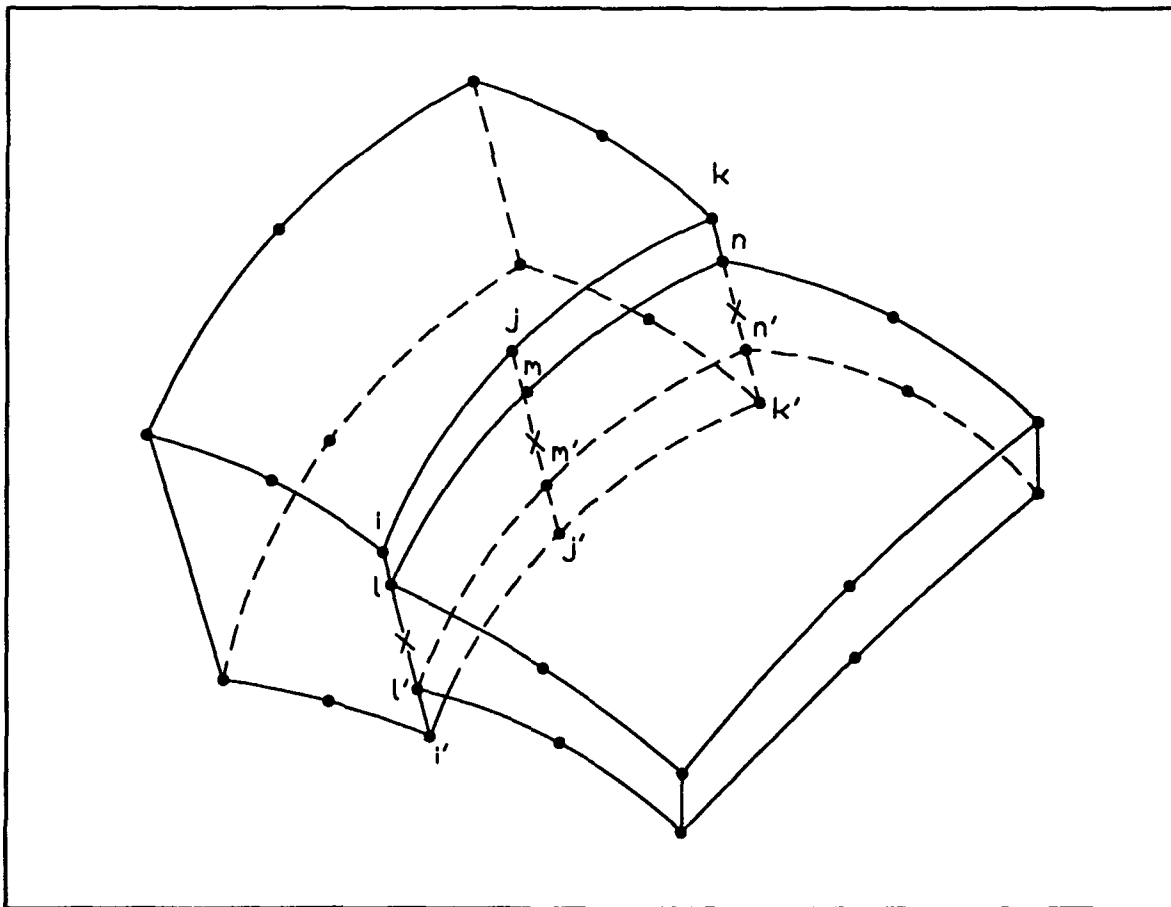


Connection of Thick-Shell Elements of Different Thicknesses

## G. 3-D (8-NODE) BRICK ELEMENT DATA

This section is not required in dynamic analysis with restart option. Otherwise, the following records are needed when 3-D (8-Node) brick elements are used in the FE model.

### Record G.1 - Control Data

| | |
|---|---|
| **MTYPE** | Element type number: enter 1 for 3-D (8-Node) brick elements. |
| **NBRK8** | Total number of 3-D (8-Node) brick elements. Enter zero if mesh generation is used. |
| **NMAT** | Number of different material types. |
| **NLD** | Number of different surface loads. Enter zero if mesh generation is used. |

### Record G.2 - Modulus of Elasticity and Poisson's Ratio

| | |
|---|---|
| **N** | Material identification number. |
| **ISOT** | = 0, for isotropic material.<br>= 1, for orthotropic material. |
| **EE(1)** | Modulus of elasticity $E_{xx}$. |
| **EE(2)** | Modulus of elasticity $E_{yy}$.* |
| **EE(3)** | Modulus of elasticity $E_{zz}$.* |
| **EE(4)** | Poisson's ratio $v_{xy}$. |
| **EE(5)** | Poisson's ratio $v_{yz}$.* |
| **EE(6)** | Poisson's ratio $v_{zx}$.* |

---

* Enter zero for isotropic material.

## Record G.3 - Shear Modulus and Thermal Coefficients

EE(7)          Shear modulus $G_{xy}$.[*]

EE(8)          Shear modulus $G_{yz}$.[*]

EE(9)          Shear modulus $G_{zx}$.[*]

EE(10)         Coefficient of thermal expansion $\alpha_x$.

EE(11)         Coefficient of thermal expansion $\alpha_y$.[*]

EE(12)         Coefficient of thermal expansion $\alpha_z$.[*]

EE(13)         Weight density of the material.


## Record G.4 - Surface Loads

This record is not needed when mesh generation is used.

N              Surface load identification number.

KTYPE          Surface pressure type:
               = 1, uniform pressure;
               = 2, hydrostatic pressure.

PR             Pressure value if KTYPE = 1.
               Weight density of water if KTYPE = 2.

ZREF           Z-coordinate of the water level. Enter zero for KTYPE = 1.

NFACE          Element face number upon which pressure acts (Figure 6.3).


## Record G.5 - Reference Temperature and Gravity Acceleration

REFT           Stress-free temperature.

GRAV           Gravity acceleration.

---

[*] Enter zero for isotropic material. It is set to $E_{xx}/2(1+\nu_{xy})$ by the program.

## Record G.6- Element Data

This record set is not needed if mesh generation is used. 3-D brick elements are numbered from 1 to NBRK8. One record is required for each element except for those that are to be generated.

NEL    Element number.

NP(1)   Node - 1.

 .

 .

 .

NP(8)   Node - 8.

NINT   Integration order.

MAT   Material number.

INC    Generation parameter.

MLD   Surface pressure number.

ISP(1)   Stress point number 1:
      Set to zero to calculate stresses at the center of the element.

ISP(2)   Stress point number 2: Set to a prescribed element face number to calculate stresses at the center of that face. If zero, only stresses at ISP(1) are calculated.
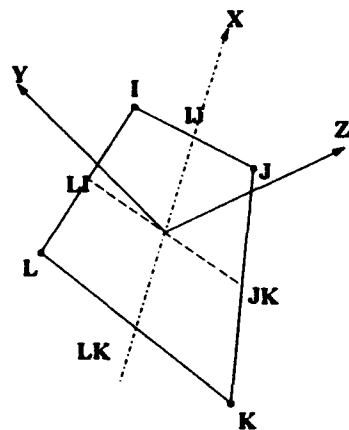
| FACE NUMBER | CORNER NODAL POINTS | | | |
|---|---|---|---|---|
| | I | J | K | L |
| 1 | 1 | 2 | 6 | 5 |
| 2 | 4 | 3 | 7 | 8 |
| 3 | 2 | 3 | 7 | 6 |
| 4 | 1 | 4 | 8 | 5 |
| 5 | 5 | 6 | 7 | 8 |
| 6 | 1 | 2 | 3 | 4 |
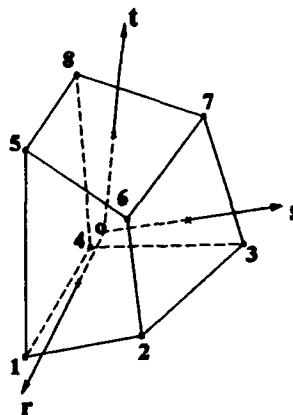
Local Stress Axes

Figure 6.3 Node and Element Face Numbering and Local Stress Axes
of 3-D Brick Element

## H. 3D-SHELL ELEMENT DATA

Skip this section for a dynamic response calculation with the restart option. Otherwise, the following records are supplied if 3-D shell elements are used in the FE model.


### Record H.1 - Control Data

| MTYPE | Element type number: Enter 2 for 3-D shell. |
|---|---|
| N3DEL | Total number of 3-D shell elements. Enter zero if mesh generation is used. |
| NMAT | Number of material types. |
| NLD | Number of surface load types. Enter zero if mesh generation is used. |


### Record H.2 - Material Properties

| MAT | Material identification number. |
|---|---|
| EE | Modulus of elasticity. |
| ENU | Poisson's ratio. |
| RHO | Weight density of material. |
| ALPT | Coefficient of thermal expansion. |


### Record H.3 - Surface Loads

This record not needed when mesh generation is used.

| N | Surface pressure identification number. |
|---|---|
| KTYPE | Surface pressure type:<br>= 1, uniform pressure;<br>= 2, hydrostatic pressure. |
| PR | Pressure value if KTYPE = 1.<br>Weight density of water if KTYPE = 2. |
| ZREF | Z-coordinate of the water level. Enter zero if KTYPE = 1. |
| NFACE | Element face number upon which pressure acts (Figure 6.4). |

## Record H.4 - Reference Temperature and Gravity Acceleration

**REFT**        Stress free temperature.

**GRAV**        Gravity acceleration.


## Record H.5 - Element Data

Two records are required for each element except for those that are to be generated. Skip this record set if mesh generation is used.


### Record H.5.1

**NEL**        Element number.

**NINT**        Integration order: = 3, for regular shape; = 4, for irregular shape.

**MAT**        Material type number.

**INC**        Generation increment.

**MLD**        Surface pressure number.

**IGG**        = 0, for 16-node elements.
                      = 1, for 12-node degenerated elements.


### Record H.5.2 - Element Connectivity

**NP(i)**        Element node numbers, i=1, 2, ..., 16.
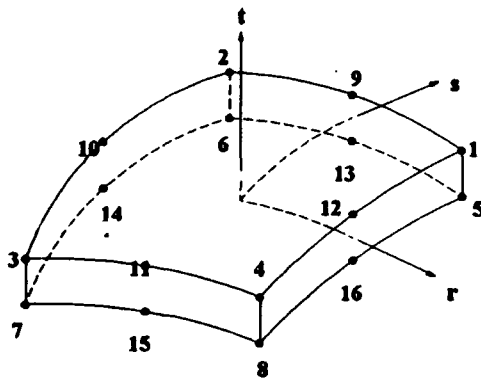
(a) 16 -Node 3-D Shell Element
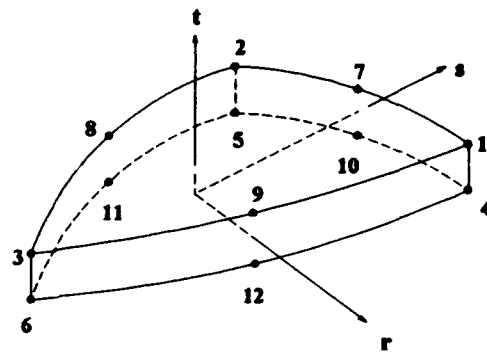
(b) 12-Node Degenerated 3-D Shell Element

| FACE NUMBER | CORNER NODAL POINTS | | | |
|---|---|---|---|---|
| 1 | 1 | 4 | 8 | 5 |
| 2 | 2 | 3 | 7 | 6 |
| 3 | 5 | 6 | 2 | 1 |
| 4 | 8 | 7 | 3 | 4 |
| 5 | 1 | 2 | 3 | 4 |
| 6 | 5 | 6 | 7 | 8 |

(c) Element Face Numbering
of 3-D Shell Element

Figure 6.4 Element Node and Face Numbering of 3-D Shell Element

# I. THICK-SHELL ELEMENTS

Skip this section for a dynamic response analysis using the restart tapes. Otherwise, the following data records should be supplied if thick-shell elements are used in the FE model.

## Record I.1 - Control Data

**MTYPE**     Element type number: enter 3 for thick-shell elements.

**NUMEL**     Total number of thick-shell elements:   Enter zero if mesh generation is used.

**NMAT**     Number of material types.

**NLD**     Number of surface load types:   enter zero, it is set by the program.

## Record I.2 - Material Properties

**MAT**     Material identification number.

**EE**     Modulus of elasticity.

**NU**     Poisson's ratio.

**RO**     Mass density of the material.

**GRAV**     Weight density of the material.

**THERM**     Coefficient of thermal expansion.

## Record I.3 - Water and Temperature Data

**ROWATER**     Weight density of water.

**REFT**     Stress-free temperature.

## Record I.4 - Element Data

Skip this record set if mesh generation is used. Otherwise, for each element two records are required, and they are numbered in increasing sequence.

### *Record I.4.1 Connectivity Data (Figure 6.5)*

NN     Element number.

IX(1)     Node 1.

 .

 .

 .

IX(8)     Node 8.

### *Record I.4.2 - Material and Pressure Types*

MAT     Material identification number.

PRESS(1)     Uniform pressure (normal) acting on face $t = -1$.

PRESS(2)     Uniform pressure (normal) acting on face $t = +1$.

(a) 16-Node Thick-Shell Element



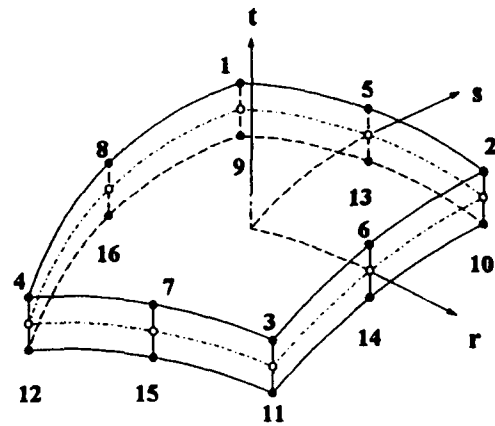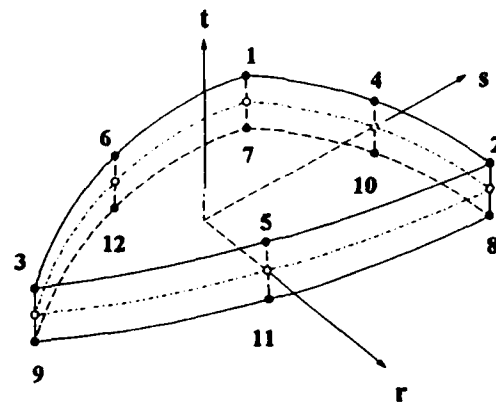(b) Degenerated Thick-Shell Element

Figure 6.5  Element Node Numbering of Thick-Shell Element

45

# J. PRE-PROCESSOR INPUT DATA DESCRIPTION

The input data for the graphics pre-processing are entered according to the specifications described in this section. The following data are required when NDYN = 4 (Record B.1).

## Record J.1 - Control Data

IPLOT        = 1, * Generates XYZ3-D plot files of the dam and foundation with hidden lines removed. The following three files are generated:

| File-name | Description |
|-----------|-------------|
| *damfn.xyz* | Complete dam and foundation model |
| *fn.xyz* | Foundation model alone |
| *halfdfn.xyz* | Right-half portion of the dam and foundation model |

Use these files as input to XYZ3-D to produce the desired 3-D pictures. The "p-7" optional parameter of the PICTURE directive of XYZ3-D can be set to 3 to save the final pictures in ASCII format. The "XYZ2-DXF" translator can then be used to convert these ASCII picture files into "DXF" format for interfacing with AutoCAD.

= 2, Generates AutoCAD 3-D and/or 2-D plot files (see Record J.4 ).

= 3, *Generates both the XYZ3-D and AutoCAD plot files.

IRES        If not zero, a prismatic FE reservoir model consisting of IRES fluid layers is generated (Figures 3.9 and A.13). The generated input data are stored in *"incres.in"* file which is used as input to the INCRES program for calculating the incompressible added mass.

## *Record J.1.1 - Shrink Plots*

This record is needed for generating XYZ3-D shrink plots, when IPLOT = 1, or 3.

GAMA        Shrinkage Factor: If zero, no shrink plot is generated. Otherwise, all dam and foundation elements are shrunk by GAMA value (less than 1) to produce 3-D shrink plots. For example, a GAMA of 0.2 will shrink each element by 20%. The AutoCAD shrink file is saved in "shrink.dxf," and the XYZ plot file is saved in "shrink.xyz" file which may be processed as described in Section J.1.

---

* XYZ3-D provides 3-D plots with hidden lines removed. It is a good supplement to AutoCAD Release 9 and lower, which did not include advanced 3-D capabilities. If you have AutoCAD Release 10 and higher, XYZ3-D is not required and you may use IPLOT = 2 only.

## Record J.2 - 8-Node Elements

This record is needed when the mesh generation is __not used__.

| | |
|---|---|
| **N3DDAM** | Number of brick elements used in the dam. |
| **N3DFN** | Number of brick elements used in the foundation. |

## Record J.3

This record is needed when IPLOT = 1 or 3, and the mesh generation is __not used__.

| | |
|---|---|
| **N3DDH** | Number of brick elements in the right-half portion of the dam. |
| **N3DFH** | Number of brick elements in the right-half portion of the foundation. |
| **NSHLH** | Number of 3-D shell elements in the right-half portion of the dam. |
| **NTSHH** | Number of thick-shell elements in the right-half portion of the dam. |

## Record J.4 - Control Data For AutoCAD Plot Files

This record is needed when IPLOT = 2 or 3.

**K3D**      If not zero, an AutoCAD plot file of the combined dam-foundation model is generated. In the generated AutoCAD "DXF" file, each element type is stored in a separate layer as follows:

| File name | Layer No. | Description |
|---|---|---|
| *mesh3d.dxf* | 1 | Foundation brick elements |
| | 2 | Dam 3-D shell elements |
| | 3 | Dam thick-shell elements |
| | 4 | Dam brick elements |

**KUS**      If not zero, a plot of the U/S face of the dam projected on a vertical plane is generated. Element and node numbers are stored on separate AutoCAD layers for easy inclusion or omission.

| File name | Layer name |
|---|---|
| *us.dxf* | NODE NUMBERS, ELEMENT NUMBERS |

**KDS**      Same as KUS, except it is for the downstream face.

| File name | Layer name |
|---|---|
| *ds.dxf* | NODE NUMBERS, ELEMENT NUMBERS |

**KLOC**          If not zero, crown cantilever together with line of centers are generated and saved in "*loc.dxf*" file. Enter zero when mesh generation is not used.

**KCANT**         If not zero, cantilever and foundation sections are saved in "*cant.dxf*" file for plotting. In this case, KCANT is the number of cantilevers to be plotted. The cantilevers for which plotting is requested are specified in Record J.5.

**KARCH**         If not zero, arch sections at each mesh elevation are plotted. The file name for arch sections is "*arch.dxf*".

**HT**          Height of a typical dam element. This is used to compute an appropriate text size for printing node and element numbers on the mesh plots. If set to zero, a default value of 30.0 units consistent with the length units of the dam model will be used.

## Record J.5 - Cantilever Sections

The following records are repeated KCANT times to specify all cantilever sections for which plotting is required. Omit if KCANT = 0. Record J.5 is needed when mesh generation is used. Otherwise, the following two records should be submitted.

**ICAN**         Cantilever number (See Figure 6.6).

**ISIDE**        Side identification:
= 1, right abutment;
= 2, left abutment.

The following two Records (J.5.1 and J.5.2) are needed when mesh generation <u>is not used</u>.

### *Record J.5.1*

**NEL**         Number of elements or polygons in the cantilever section.

### *Record J.5.2*

This record specifies the element or polygon node numbers in a sequential manner. Repeat this record for all NEL elements.

**NNOD**        Number of element or polygon nodes.

**IN(i)**         Element or polygon node numbers 1, 2, ..., NNOD.
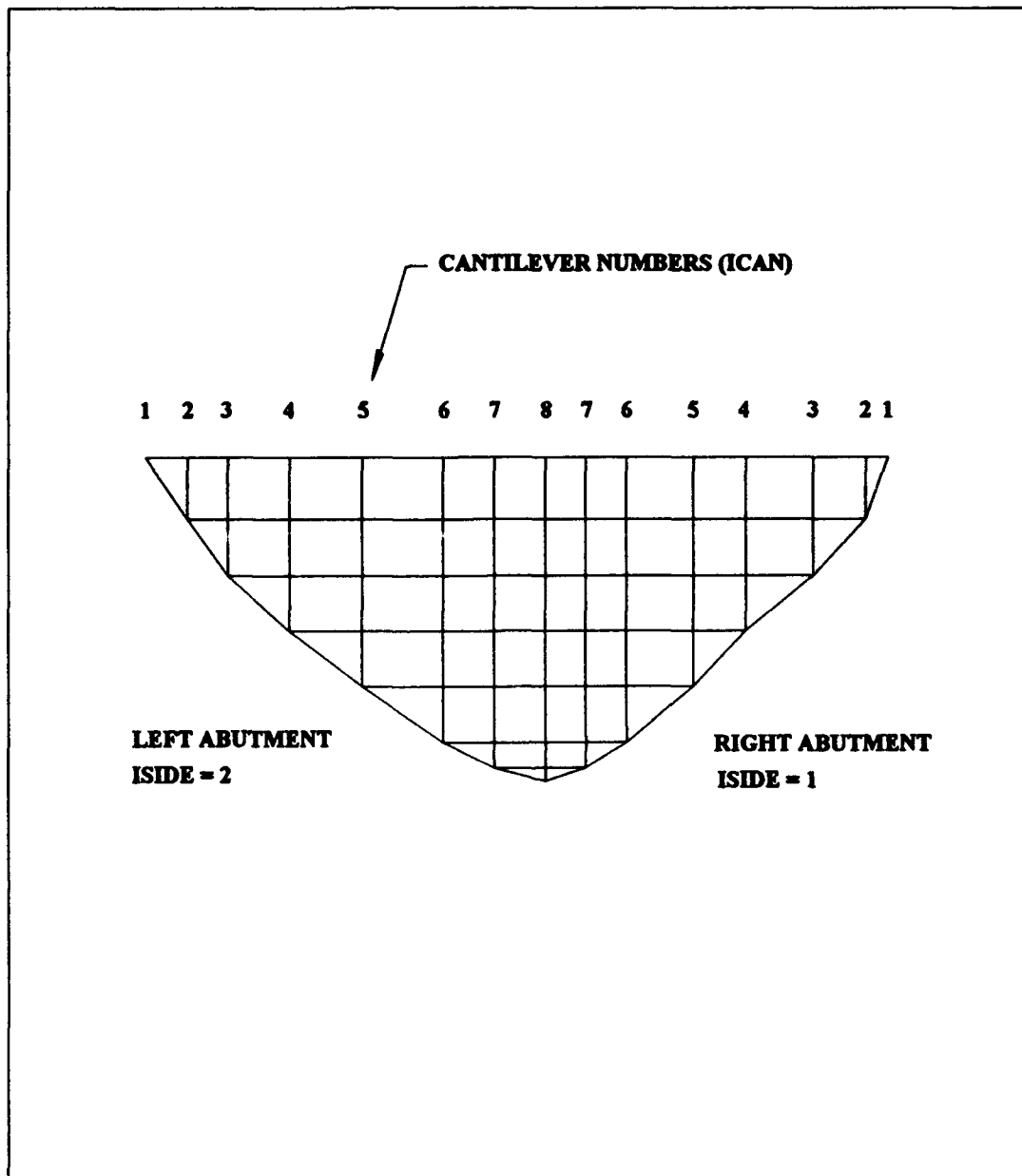
Figure 6.6   Cantilever Numbers

49

## Record J.6 - Arch Sections

This record is needed when a plot of arch sections has been requested (i.e. KARCH is not zero), and the mesh generation is not used.

NELV                 Number of mesh elevations.

ELM(1)               Mesh elevation at the base of the dam.

.
.

ELM(NELV)            Mesh elevation at the crest.


## Record J.7 - Reservoir Mesh

When IRES (see Record J.1) is not zero, records J.7.1 and J.7.2 are needed to generate an FE mesh for the reservoir. In addition, "*fluid3d.in*", a preparatory fluid data file, should also be available (see Section 1-A of the Appendix).

### *Record J.7.1*

WATL        Reservoir water level for added-mass calculation (i.e. Z-coordinate).

Z2          Z-coordinate of the upper edge of the dam elements that have their upper edges just above the water surface or coincide with the water surface.

Z0          Z-coordinate of the midheight of the dam elements described above; enter zero, if WATL=Z2.

Z1          Z-coordinate of the lower edges of the corresponding dam elements; enter zero, if WATL=Z2.


### *Record J.7.2*

The following records are needed when water surface does not coincide with a mesh elevation containing concrete element nodes. In that situation, the coordinates of the interface nodes at the water surface and immediately below it should be provided.

NODE        Node number.
X           X-coordinate.
Y           Y-coordinate.
Z           Z-coordinate.

Provide as many records as necessary until all surface nodes and nodes immediately below them have been specified.

## K. STATIC ANALYSIS

The following records are required in static analysis only.

### Record K.1 - Concentrated Nodal Loads

For each nodal point at which concentrated forces or moments are applied, a number of records are required. This number is equal to the number of load cases (LL in Record B.1) in which concentrated loads are acting at that nodal point. The data records are provided according to the nodal number sequence and should be terminated by a record containing zero in each data field. Each record contains the following information:

N               Node number.

L               Load case number.

R(1)            Force in X-direction.

R(2)            Force in Y-direction.

R(3)            Force in Z-direction.

R(4)            Moment about local x-axis.

R(5)            Moment about local z-axis.


### Record K.2 - Reaction Force Control Data

Reaction forces (arch thrusts) at any desired interface nodes with the foundation rock (Figure 6.7), or at any desired contact nodes with a gravity thrust block (Figure 6.8) may be calculated by specifying the following information.

NUMRE           Number of reaction elements.

NUMRN           Number of reaction nodes.


NUMRE and NUMRN should be set to zero, when computation of reaction forces are not required. In that case the following two records (K.2.1 and K.2.2) should be skipped.

## Record K.2.1 - Reaction Element Definition

**IELEM(i)**      Reaction element numbers, i=1,2,...NUMRE.


## Record K.2.2 - Reaction Node Definition

**INODE(i)**      Reaction node numbers, i=1,2,...NUMRN.


## Record K.3 - Element Loads

For each load case, one record is supplied to specify the element loads to be considered in the analysis. Each load multiplier as defined below can be used to include or exclude any of the three basic load types. A nonzero load multiplier can also be used to scale the corresponding load. For example, an AA = 1.2, which increases the gravity loads by 20%, is equivalent to increasing the unit weight of the concrete by 20%. This way the input data need not to be changed except for this record. There are a total of LL load cases as specified in Section B.1.


**AA**            Gravity load multiplier:
                  $\neq 0$, include gravity load and scale by AA;
                  $= 0$, exclude gravity load.

**BB**            Water load multiplier, same as AA.

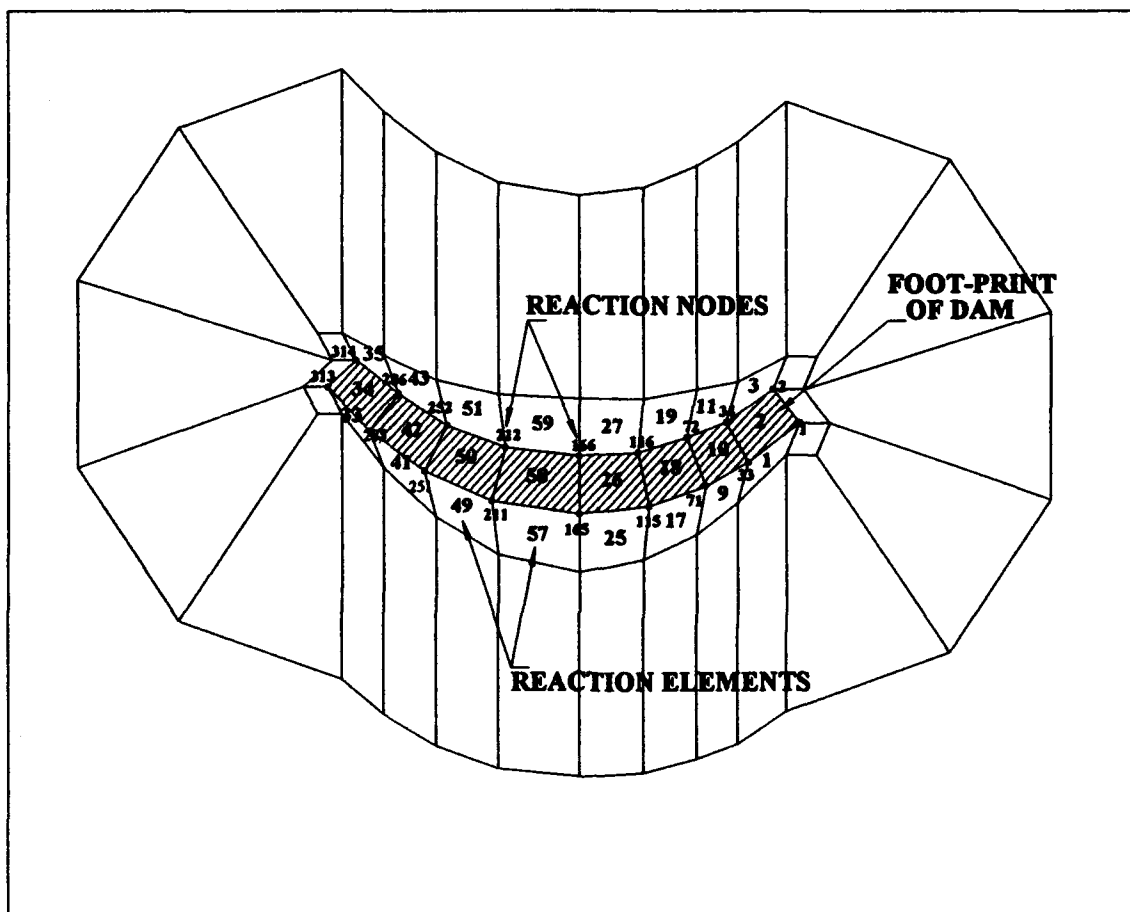**CC**            Temperature load multiplier, same as AA.

Figure 6.7  Plan View of Foundation Model Showing Dam-Foundation Reaction Nodes and
Elements

Reaction Nodes:  1, 2, 23, 34, 71, 72, 115, 116, 165, 166, 211, 212, 251, 252, 285,
286, 313, 314

Reaction Elements:  1, 2, 3, 9, 10, 11, 17, 18, 19, 25, 26, 27, 57, 58, 59, 49, 50, 51,
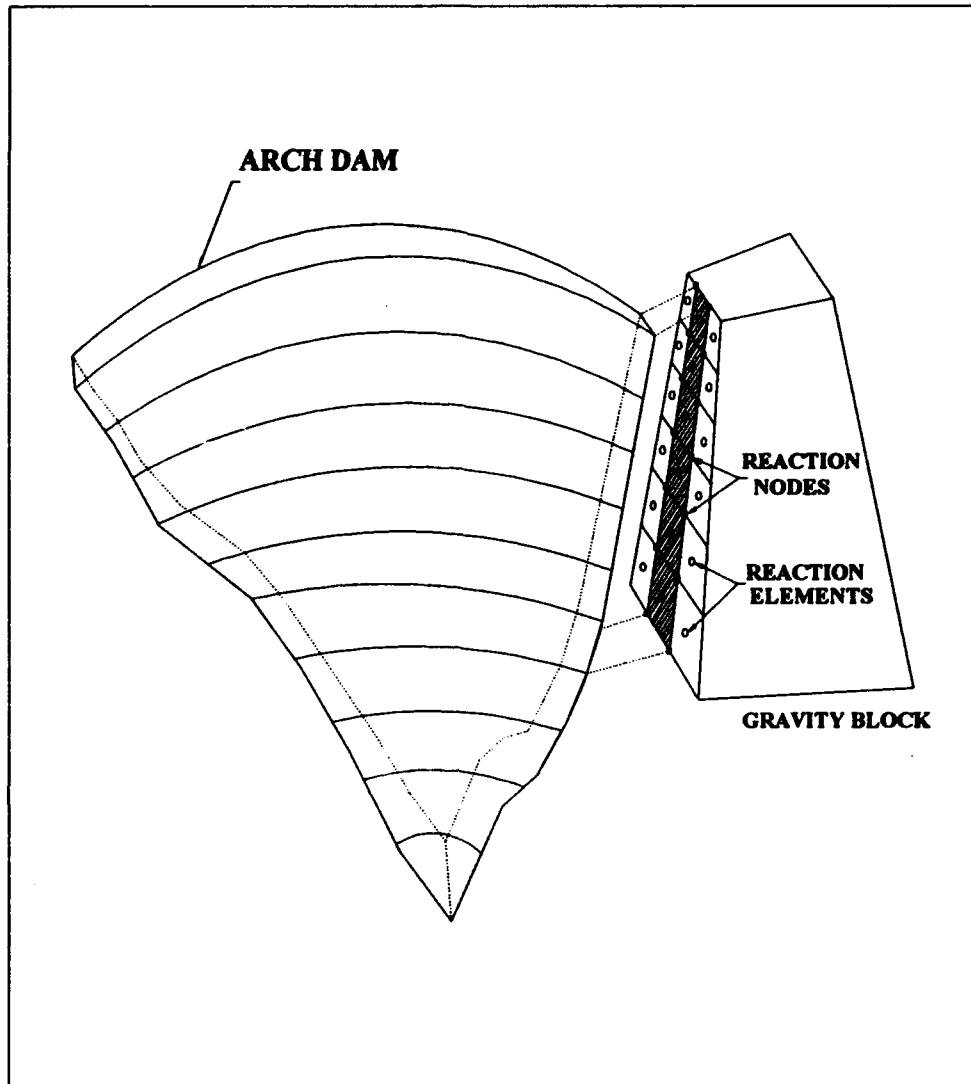41, 42, 43, 33, 34, 35

Figure 6.8  Dam-Gravity Block Reaction Nodes and Elements

## L. TIME-HISTORY ANALYSIS

The following records are needed in response-history analysis only.

### Record L.1 - Response Control Data

| NFN | Number of components of ground motion. |
|---|---|
| DT | Integration time step. * |
| NT | Total number of analysis time-steps. |
| NOT | Time interval for printout of nodal displacements and stresses, expressed as a multiple of the integration time-step. |
| DAMP | Modal damping ratio to be applied to all modes. |

### Record L.2 - Ground Motion Control Data

| JFN(1) | Identification number for the ground motion in the x-direction. |
|---|---|
| JFN(2) | Identification number for the ground motion in the y-direction. |
| JFN(3) | Identification number for the ground motion in the z-direction. |

### Record L.3 - Ground Motion

The following set of records is required for each component of the ground motion. The sequence should correspond to ground motion identification numbers and in increasing order.

### *Record L.3.1 - Control Data*

| NLP | Number of acceleration data points. |
|---|---|
| SFTR | Scale factor multiplier (default = 1.0). It is also used to convert the input accelerations into appropriate units. |

---

* The same integration time-step is specified for all modes. In general, a time-step at least 5 to 10 times less than the lowest period of vibration will provide good accuracy for all modes that are considered in the analysis. To assure numerical stability and accuracy of the solution, the GDAP program automatically filters the high mode response, for which the period of vibration is less than 5 times the integration step.

*Record L. 3.2 - Header*

**HED**        Title for the input motion.


*Record L. 3.3 - Acceleration Data*

**T**        Time value at point 1.

**P**        Acceleration value at point 1.

...        ...

Four pairs of time and acceleration values are supplied in each record. As many records as required are provided to specify NLP pairs of data points.


## Record L.4 - Displacement Output

The following set of records is required to specify the displacement output results.


*Record L. 4.1 - Control Data*

**KKK**        Code for output type:
                = 1, printout of displacement histories and maxima;
                = 2, plot of displacement histories and printout of maxima;
                = 3, printout of displacement maxima only.

**ISP**        Plot spacing indicator.


*Record L. 4.2 - Displacement Components*

One record is required for each nodal point for which displacement plots or printout is requested. The set of records is in increasing order of nodal numbers. One record consisting of six zeros is supplied to terminate the sequence of records. Up to five displacement components may be requested for the thick-shell nodes and up to three components for all other nodes.

**NP**        Node number.

**IC**        Displacement component:
.             = 1, X-component;
.             = 2, Y-component;
.             = 3, Z-component;
.             = 4, local x-rotation;
              = 5, local z-rotation.

## Record L.5 - Stress Output

The following records are required to specify the stress output.

### Record L.5.1 - Control Data

KKK   Code for output type:
       = 1, printout of stress histories and maxima;
       = 2, plot of stress histories and printout of maxima;
       = 3, printout of stress maxima only.

ISP    Plotting interval.

### Record L.5.2 - Stress Components

For each element type used, a set of two records is required to specify the requested stress components. The order of 3-D brick, 3-D shell, and thick shell should be followed. For each element, the first record contains desired element number (NEL) and its associated number of stress components (NCOMP); and the second record specifies NCOMP requested stress components. Each set is terminated by a record with two zeros for NEL and NCOMP as described.

### Record -1:

NEL   Element number.

NCOMP  Number of requested stress components.

### Record -2:

This record contains the requested stress components of the element specified in Record-1. Up to 12, 60, and 40 stress components may be requested for 3-D brick, 3-D shell, and thick-shell elements, respectively. The complete list of stress components for each element type is summari~ed in the following tables and also shown in Figures 6.9 and 6.10.

Although all stress components of all elements in the structure can be specified for output, only arch, cantilever, and the shear stresses in the surface direction are often used. Accordingly, the GDAP postprocessor only accepts arch, cantilever, and the surface shear stresses at each stress point. Therefore, for postprocessing purposes only these three stress components are considered and are prescribed according to the following rules.

1. 8-node Solid: Stress components are specified only for the 8-node elements used in the dam. For each element two stress points, one at the center of one face and another at the centroid of the elements are specified (Table 6.1).

2. 3-D Shell: Specify arch, cantilever, and the surface shear stress components at all stress points and for all elements, except at the midedge locations common with the adjacent 3-D shell elements (Table 6.2).

3. Thick-shell: Specify arch, cantilever, and the surface shear stress components at all eight stress points for all elements (Table 6.3).

### Table 6.1 Stress Components in 8-Node Solid Elements

| Stress Components | Face Center / or Centroid, Point 1 | Face Center Point 2 |
|---|---|---|
| $\sigma_{xx}$ | 1 | 7 |
| $\sigma_{yy}$ | 2 | 8 |
| $\sigma_{zz}$ | 3 | 9 |
| $\sigma_{xy}$ | 4 | 10 |
| $\sigma_{yz}$ | 5 | 11 |
| $\sigma_{zx}$ | 6 | 12 |

### Table 6.2 Stress Components in 3-D Shell Elements

| Stress Components | Stress Points |
|---|---|

| Stress Components | 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
|---|---|---|---|---|---|---|---|---|---|---|
| $\sigma_{xx}$ | 1 | 7 | 13 | 19 | 25 | 31 | 37 | 43 | 49 | 55 |
| $\sigma_{yy}$ | 2 | 8 | 14 | 20 | 26 | 32 | 38 | 44 | 50 | 56 |
| $\sigma_{zz}$ | 3 | 9 | 15 | 21 | 27 | 33 | 39 | 45 | 51 | 57 |
| $\sigma_{xy}$ | 4 | 10 | 16 | 22 | 28 | 34 | 40 | 46 | 52 | 58 |
| $\sigma_{yz}$ | 5 | 11 | 17 | 23 | 29 | 35 | 41 | 47 | 53 | 59 |
| $\sigma_{zx}$ | 6 | 12 | 18 | 24 | 30 | 36 | 42 | 48 | 54 | 60 |

### Table 6.3 Stress Components in Thick-Shell Elements

| Stress Components | 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 |
|---|---|---|---|---|---|---|---|---|
| $\sigma_{xx}$ | 1 | 6 | 11 | 16 | 21 | 26 | 31 | 36 |
| $\sigma_{yy}$ | 2 | 7 | 12 | 17 | 22 | 27 | 32 | 37 |
| $\sigma_{xy}$ | 3 | 8 | 13 | 18 | 23 | 28 | 33 | 38 |
| $\sigma_{yz}$ | 4 | 9 | 14 | 19 | 24 | 29 | 34 | 39 |
| $\sigma_{zx}$ | 5 | 10 | 15 | 20 | 25 | 30 | 35 | 40 |
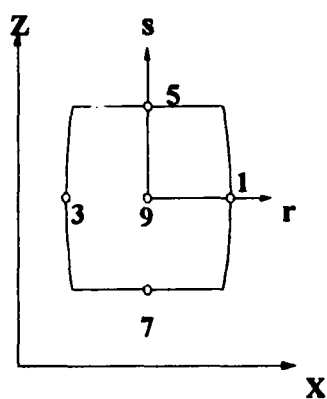
(a) Upstream Face
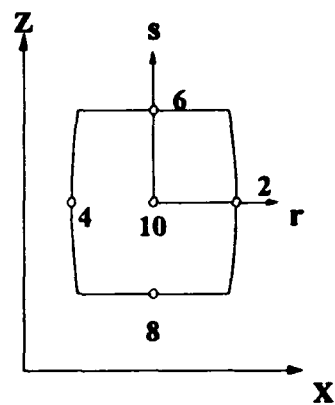
(b) Downstream Face

| Stress Point | r | s | t | Corresponding Node No. | |
|---|---|---|---|---|---|
| | | | | 16-Node Element | 12-Node Element |
| 1 | +1 | 0 | +1 | 12 | 9 |
| 2 | +1 | 0 | -1 | 16 | 12 |
| 3 | -1 | 0 | +1 | 10 | 8 |
| 4 | -1 | 0 | -1 | 14 | 11 |
| 5 | 0 | +1 | +1 | 9 | 7 |
| 6 | 0 | +1 | +1 | 13 | 10 |
| 7 | 0 | -1 | -1 | 11 | NONE |
| 8 | 0 | -1 | +1 | 15 | NONE |
| 9 | 0 | 0 | -1 | | |
| 10 | 0 | 0 | +1 | | |

(c) Location of Stress Points

Figure 6.9  Stress Point Locations of 3-D Shell Element

59

(a) Upstream Face　　　(b) Downstream Face

| Stress Point | r | s | t |
|---|---|---|---|
| 1 | -0.5774 | -0.5774 | -1 |
| 2 | -0.5774 | -0.5774 | +1 |
| 3 | -0.5774 | +0.5774 | -1 |
| 4 | -0.5774 | +0.5774 | +1 |
| 5 | +0.5774 | -0.5774 | -1 |
| 6 | +0.5774 | -0.5774 | +1 |
| 7 | +0.5774 | +0.5774 | -1 |
| 8 | +0.5774 | +0.5774 | +1 |

(c) Location of Stress Points
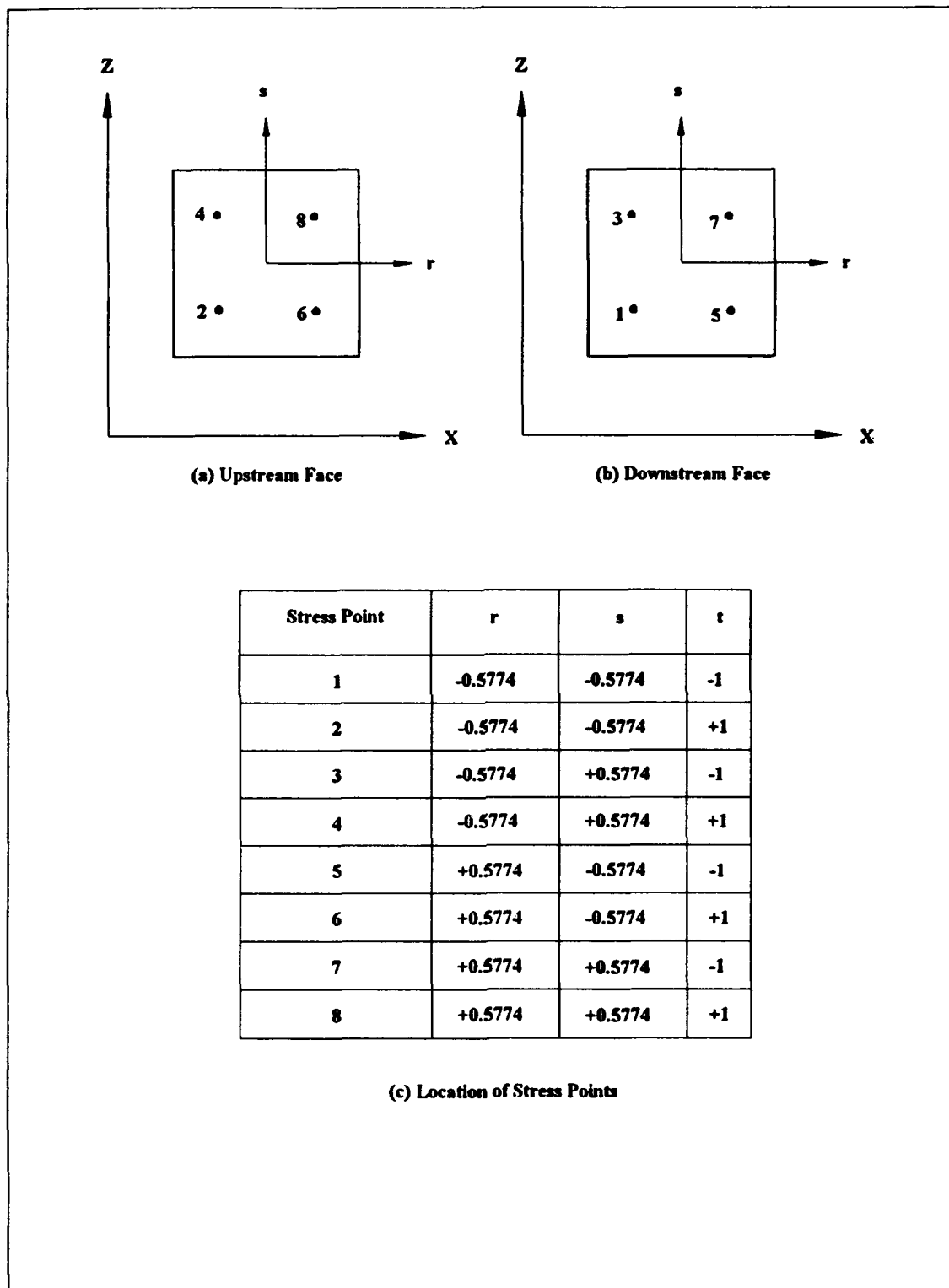
Figure 6.10  Stress Point Locations of Thick-Shell Element

## M. RESPONSE-SPECTRUM ANALYSIS

The following records are required only in response-spectrum analysis:

### Record M.1 - Control Data

| | |
|---|---|
| **MGM** | Number of components of the ground motion (1, 2, or 3). |
| **IPD** | Code for displacement output:<br>= 1, printout of modal and SRSS displacements are desired;<br>= 0, no displacement printout. |
| **IPS** | Code for stress output:<br>= 0, compute stresses, print, and save SRSS stresses for plotting;<br>= 1, do not compute stresses. |
| **IPMOD** | Code for modal stress output (IPS = 0):<br>= 0, no printout of modal stresses;<br>= 1, print modal stresses for all elements. |

### Record M.2 - Acceleration-Spectrum Data

The following set of records should be supplied for each component of the ground motion (follow the X, Y, and Z order). If no ground motion is to be considered in a particular direction, two records with two zeros should be substituted instead.

#### Record M.2.1 - Header Information

| | |
|---|---|
| **HED** | Title of ground motion. |

#### Record M.2.2 - Control Data

| | |
|---|---|
| **NP** | Number of points specifying the acceleration spectrum. |
| **SFTR** | Scale factor; use to scale spectral accelerations or to convert them into consistent units. |

#### Record M.2.3 - Response-Spectrum Data

| | |
|---|---|
| **T** | Period value at point 1. |
| **S** | Acceleration value at point 1. |
| ... | ... |

One to four pairs of period and acceleration spectrum are specified in each record. Supply as many records as required to define all NP points. Linear interpolation is used in the program to calculate spectrum values for the periods between the specified input points.

# 7. INCRES INPUT DATA DESCRIPTION

The input data for calculating the FE added mass for the incompressible water is generated by the GDAP pre-processor according to the specifications described in this section. The pre-processor generates a standard prismatic reservoir water mesh extending upstream to a distance equal to the number of fluid layers multiplied by the water depth (IRES*WATL). The number of fluid layers IRES (Record J.1) is normally set to 3 or 5 as shown in Figures 3.9 and A.13. The liquid nodes are obtained by projecting the concrete nodes in the upstream direction. The reservoir boundaries and the upstream vertical plane are assumed to be rigid. For reservoirs with complicated topography, the standard mesh described above may not be adequate. In that case, the generated input data may be modified or the entire input data may be manually created to represent the actual topography of the reservoir.

## Record A - Title

This record contains information to be printed as the output header.

## Record B - Control Data

| | |
|---|---|
| NUMNP | Total fluid nodal points. |
| NUMNS | Number of fluid nodal points on the dam-reservoir interface. |
| N3DEL | Number of 3-D fluid elements. |
| N2DEL | Number of 2-D interface fluid elements. |
| WMASS | Mass density of water. |
| GA | Gravity acceleration. |
| WATL | Z-coordinate of water level. |
| ICOMP | Comparison between the FE and Westergaard solution: * |
| | EQ. 0, no comparison is made; NE. 0, a comparison is made by subjecting the dam face to a pattern of unit g uniform accelerations in the ICOMP direction. (ICOMP= 1, 2, or 3 corresponding to the X, Y, and Z ). |
| IPLOT | Code for generation of plot files: |
| | = 0, do not generate plot file, solve for added mass; |
| | < 0, generate AutoCAD and XYZ3-D plots; |
| | > 0, generate AutoCAD plot only. |

---

* The comparison between the FE and the Westergaard method is made only for a simple pattern of unit g accelerations that are applied in the global X, Y, or Z direction. This is essentially equivalent to a rigid body motion of the dam, and thus flexibility of the arch structure is not considered. The resulting hydrodynamic pressures acting on the face of the dam and the equivalent nodal forces for each method are printed out in the output file.

## Record C - Nodal Coordinates

One record per nodal point is required to specify the coordinates and the boundary conditions. The sequence of node numbering on each liquid section must follow the order in which the concrete nodes on the interface have been numbered.

| | |
|---|---|
| N | Node number. |
| XYZ(1) | X-coordinate. |
| XYZ(2) | Y-coordinate. |
| XYZ(3) | Z-coordinate. |
| IBC | Boundary conditions: |

EQ. 0, noninterface nodes below the water surface.

EQ. 1, noninterface nodes at the water surface.

EQ. -1, interface nodes below the water surface.

EQ. -2, interface nodes at the water surface.

## Record D - 2-D Element Data

One or both of the following records are required for each 2-D element on the interface. The sequence of data is in increasing order of the absolute values of the element numbers.

### D.1 - Element Connectivity

This record is always required. The element nodes must be input according to the order indicated in Figure 7.1a.

| | |
|---|---|
| NEL | Element number: |
| | For an element that its nodes at the water surface do not coincide with the corresponding concrete nodes, NEL is entered as a negative number. |
| NCON(1) | Element nodal point 1.* |
| ... | ... |
| NCON(8) | Element nodal point 8. |
| NINT | Integration order: 2 or 3 (2 is usually adequate). |

---

* Degenerated nodal points and the omitted midedge nodes should be set to zero. For example, the element connectivity for the triangular 2-D element in Figure 7.1a is 1, 2, 3, 0, 5, 6, 0, 8.

## D.2 - Z-coordinate of Surface Elements

This record is only required when NEL is negative. That is when the water surface level does not coincide with the upper concrete element nodes.


**Z2**        Z-coordinate of the upper-edge nodes of concrete elements at the water surface.

**Z0**        Z-coordinate of the midheight nodes of concrete elements at the water surface.

**Z1**        Z-coordinate of the lower-edge nodes of concrete elements at the water surface.

**Note:**

Water level always lies between Z2 and Z1.


## Record E - 3-D Element Data

Two records are required for each 3-D fluid element. The sequence of records is in increasing order of the element numbers.


### E.1 - Element Identification

**NE**        Element number.

**NINT**      Integration order: 2 or 3 (usually 2 is sufficient).


### E.2 - Element Connectivity

The element nodes must be input according to the order indicated in Figure 7.1b.


**NP(1)**     Element node number 1.*
...           ...
**NP(16)**    Element node number 16.


---

* Degenerated nodal points and omitted midedge nodes should be set to zero. For example, element numbering of the triangular 3-D element in Figure 7.1b is 1, 2, 3, 0, 5, 6, 7, 0, 9, 10, 0, 12, 13, 14, 0, 16.

## Record F - DOF's of Dam Interface Nodes

DOF's of dam nodes on the interface are entered according to the sequence of the reservoir interface nodes (which by design follow the sequence of concrete nodes). DOF's of these nodes may be obtained from the ID array of any previous GDAP analysis.

For each node only three translational DOF's are considered. Sixteen values per record are provided, and as many records as needed are supplied to specify all DOF's of all interface nodes.

(a) 2-D Interface Elements

(b) 3-D Liquid Elements
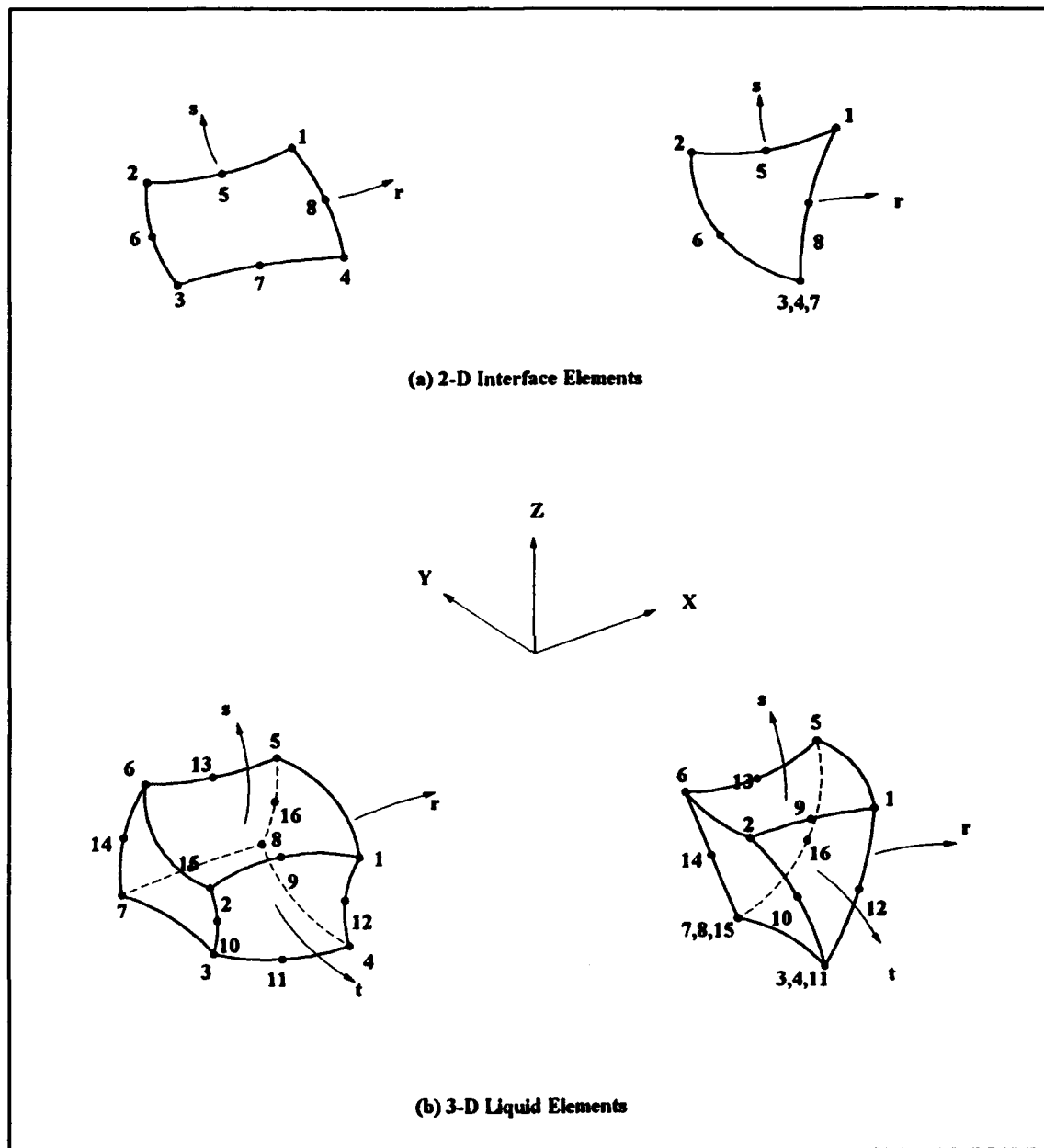
Figure 7.1  2-D and 3-D Liquid Elements

# 8. POST-PROCESSOR INPUT DATA DESCRIPTION

The GDAP post-processor is run after the completion of static or dynamic analysis. It reads files generated by GDAP during the static and dynamic runs and produces appropriate plot files for the AutoCAD and POSTPLT graphics packages. The additional input data required for running POSTPRS is described in this section.

## Record A - 8-Node Elements of the Dam

**N3DDAM**        Number of 8-node elements used in the arch.

## Record B - Control Data

**LL**        Number of load cases in static analysis or number of mode-shapes to be plotted. Enter 1, for post-processing of the dynamic response.

**KDISP**        Code for plot of static displacements. If not zero, displacement plots for LL static load cases are generated and stored in file "disps.dxf". Data for each load case are stored in a separate AutoCAD layer; layer "LOAD-1" for load case 1, and layer "LOAD-LL" for the load case LL. KDISP may be selected as a positive or negative number to represent one of the following conditions.

KDISP > 0,     displacements are plotted along horizontal arch sections only (See Record F.1 in this section).

KDISP < 0,     displacements are plotted along any continuous section (Record F.2).

**KMODE**        Code for plot of mode-shapes. If not zero, LL mode-shapes are processed and stored in "modes.dxf". Data for each mode is stored in a separate layer. For example layer "MODE-i" contains the mode i.

KMODE > 0,     Mode-shapes are plotted along horizontal sections.

KMODE < 0,     Mode-shapes are plotted along nonhorizontal sections.

**KSTAT**        Code for post-processing of static stresses. If not zero, stresses due to each load case are separated into the upstream and downstream components and are stored in separate files that are used as input to POSTPLT to prepare contour plots of the arch and cantilever stresses or vector plots of the principal stresses. The designated names of the stress files are shown in the table in Appendix A, Section D, POSTPLT Runs, bottom of p 78.

**KRSPEC**        Code for post-processing of response-spectrum stresses. If not zero, SRSS stress contour files for both faces of the dam are generated.

**KTH**        Code for post-processing of time-history results. If not zero, time-history results are post-processed.

**DSF**        Displacement scale factor (DSF); displacements are multiplied by DSF to convert into desired units.

**SSF**        Stress scale factor; stresses are divided by SSF to convert into desired units.

## Record C - 3-D Brick Elements to be Excluded

This record set specifies 3-D brick elements which are to be excluded from the displacement or stress plots.

Static displacements and mode shapes are plotted for the upstream nodes of the dam face. It is assumed that there are three brick elements through the dam thickness. Consequently, those brick elements that are not part of the arch structure or the foundation rock (elements in the spillways or other structural features) should be excluded. Similarly, the stress plots are generated for the upstream and downstream faces of the dam, and thus brick elements that are not part of the arch structure are excluded.

### Record C.1 - Number of 3-D Brick Elements

**NS1**        Number of brick elements to be excluded from the plots. Enter zero, if there are none.

### C.2 - 3-D Brick Element Numbers

**NELM1(i)**        Brick element numbers, i=1,2,...NS1.

## Record D - 3-D Shell Elements to be Excluded

This record set specifies the list of 3-D shell elements to be excluded from the displacement or stress plots. This includes all 3-D shell elements that are not part of the arch structure or do not have common surface with the upstream face of the dam.

### Record D.1 - Number of 3-D Shell Elements

**NS2**        Number of 3-D shell elements to be excluded from the plots.

_Record D.2 - 3-D Shell Element Numbers_

**NELM2(i)**    3-D shell element numbers,  i = 1,2,...NS2.


## Record E - Thick-shell Elements to be Excluded

This record specifies the list of those thick-shell elements that have no common surface with the upstream and downstream faces of the dam and should be excluded from the plots.


_Record E.1 - Number of Thick-shell Elements_

**NS3**    Number of thick-shell elements to be excluded from the plots.


_Record E.2 - Thick-shell Element Numbers_

**NELM3(i)**    Thick-shell element numbers,  i = 1,2,...NS3


## Record F - Mesh Elevations

This record is required when the plot of static displacements or dynamic mode shapes is requested. The deflections and mode shapes are plotted either along horizontal arch sections that are specified by their mesh elevations alone or along any continuous section which is specified by its nodal points and a representative elevation.

**NELV**    Number of mesh elevations for which plot of displacements or mode-shapes are required.

**PORD**    Maximum ordinate of displacement or mode-shape plots.  When set to zero, a value of 75 consistence with the length units of the model is assumed.


_Record F.1 - Elevation of Horizontal Sections_

This record is required when KDISP or KMODE > 0.

**ELM(i)**    Mesh elevations, starting at the dam base,  i = 1,2,...NELV.


_Record F.2 - Elevations and Nodal Points of Nonhorizontal Sections_

These records are needed when KDISP or KMODE < 0.  The following data should be specified for each elevation for which plot of displacement or mode shape is requested.

**ELM(i)**       Representative elevation i;  base elevation is always entered first.

**MD(i)**        Number of nodal points along elevation i.

**NODID(j)**     Nodal points along elevation i, j=1,2,...MD(i).


## Record G - Boundary Nodes

This record is needed when post-processing of static or response spectrum stresses is requested. Boundary nodes consist of the dam-foundation interface nodes and the nodes along the dam crest, and are used to define boundaries of the stress plots.


### Record G.1 - Number of Boundary Nodes

Only upstream nodes of the interface and the dam crest are required.

**NU**           Number of boundary nodes.


### Record G.2 - List of Boundary Nodes

**NUS(i)**       Boundary nodes,  i = 1,2,...NU.


## Record H - Post-processing of Time-History Results

Skip the following records if post-processing of time-history results is not desired (i.e. KTH = 0).


### Record H.1 - Control Data

**KTHDSP**       If not zero, displacement histories are processed and stored in file "thdisp.plt" which will be used as input to POSTPLT.

**KTHSTR**       If not zero, stress  histories are  processed and plot files are generated. For the name plot files (see table in Appendix A, Section D, POSTPLT Runs, bottom of p 78).

**KTHSTAT**      If not zero, static and  dynamic  stresses are combined and the post-processing is performed for the combined stresses.  If zero, dynamic stresses are not combined with the static stresses and all the subsequent post-processing are performed for the dynamic stresses alone.

## Record H.2 - Tensile Strength of Concrete

**TSTRNG**      Tensile strength or cracking strength of the concrete. This is used to identify all tensile stresses exceeding the cracking strength of the concrete.

## Record H.3 - File Name of Static Stresses

The following two records are required when KTHSTR is not zero. They contain the name of the static stress files that will be combined with time-history stresses or will be used to set up the stress plot files for the dynamic stresses alone.

## Record 1

**USNAME**      Name of the file containing the upstream stresses for the desired static load case. Example: ustat1.str, ..., ustatLL.str, or any combination of these.

## Record 2

**DSNAME**      Corresponding name for the downstream stresses.

# APPENDIX A

# EXAMPLE DAM MODEL

## ANALYSIS OF AN EXAMPLE DAM MODEL

An example dam-water-foundation model is developed to demonstrate the static and dynamic analysis capabilities of the Arch Dam Analysis Workstation. A complete analysis includes model generation using the GDAP pre-processor, static and dynamic response calculations, and post-processing of the analysis results.

The static analyses may be carried out for the separate or combined action of the various static loads. If computed separately, not only the dam response for each individual load is available, but they can also be combined by the post-processor to obtain the total response for various loading combinations. The dynamic analysis includes both response-spectrum and time-history modal superposition methods. The added mass of the incompressible water can be obtained by both the generalized Westergaard and the FE procedures.

The GDAP post-processor is used to present static deflections, vibration mode-shapes, stress contours, and vector plots of principal stresses for the separate and combined static plus dynamic loads. In the time-history analysis, envelope and the critical concurrent stresses are automatically obtained by running the GDAP post-processor and are presented in the form of stress contours for both faces of the dam. Vector plots of the principal stresses for the static plus time-history are also provided. Furthermore, time-history of input ground accelerations; time-histories of critical nodal displacements; and time-histories of all stresses exceeding the cracking strength of the concrete are presented.

The following six steps summarize analysis procedures for the example dam model. In each step, the input files, output files, and the associated programs are described. The input files in each analysis are specified by the user or obtained from the results of previous runs. The user-defined input files are **boldfaced**.

# 1. PRE-PROCESSING RUNS

## A. GDAP Runs:

Run GDAP program with **exprep.in** as input to automatically generate FE meshes for the dam, foundation rock, and reservoir water. The **fluid3d.in** is an input file needed for generating fluid elements for the reservoir model. It contains element connectivities for a limited number of fluid elements located around the boundaries of the upstream fluid layer as shown in Figure A.13. Only the 8th node corresponding to the upstream face of these elements need to be specified (see **fluid3d.in**). The GDAP input file **exprep.in** was prepared according to the description provided in the previous sections of this manual. In this example, the GDAP mesh generator is used and all the pre-processor options are activated by setting them to nonzero values. To run GDAP, at the DOS prompt type:

> **GDAP**

> Enter Input file name: **exprep.in**
> Enter Output file name: **exprep.out**

A summary of the input and output files follows:

| Program | Input Files | OUTPUT FILES | | |
|---------|-------------|--------------|----------|--------------|
| | | **DXF Files** | **XYZ Files** | **INCRES Files** |
| **GDAP** | **exprep.in** **fluid3d.in** | *mesh3d.dxf* *us.dxf* *ds.dxf* *loc.dxf* *cant.dxf* *arch.dxf* | *damfn.xyz* *fn.xyz* *halfdfn.xyz* *shrink.xyz* | *incres.in* |

## B. AutoCAD Runs:

Run AutoCAD to display various graphics files generated in Step A for examination, editing, adding captions, plotting, or generating DWG and SLIDE files (Figures A.1 to A.6).

## C. XYZ3D Runs (optional):

The last record of an XYZ input file includes the PICTURE directive or command for the XYZ3D graphics package. The "P-7" parameter for the PICTURE directive is initially set to 1 to draw and then save the generated picture on a binary file. The XYZ3D prompts you for the name of this binary file. You may use the same input file name with extension "*.plt*" (see following output files table). If you wish to generate a different view of the picture, change the coordinate of the observer's position (OBSERVER directive) in the input file and rerun XYZ3D.

In addition, you can set "P-7" to 3 and rerun XYZ3D to save the generated picture in ASCII format for interfacing with AutoCAD. When prompted for the name of this picture file, enter the same input file name with extension "*.pic*" .

## OUTPUT FILES

| Program | Input Files | (P-7 = 1) | (P-7 = 3) |
|---------|-------------|-----------|-----------|
| **XYZ3D** | *damfn.xyz* | *damfn.plt* | *damfn.pic* |
| | *fn.xyz* | *fn.plt* | *fn.pic* |
| | *halfdfn.xyz* | *halfdfn.plt* | *halfdfn.pic* |
| | *shrink.xyz* | *shrink.plt* | *shrink.pic* |

## D. XYZ2DXF Runs:

Run XYZ2DXF utility program to convert the ASCII picture files (with extension "*.pic*") generated in Step C to AutoCAD DXF files. The input and output files are specified interactively. When prompted by XYZ2DXF, enter the name of input and output files according to the following table:

| Program | Input Files | DXF Files |
|---------|-------------|-----------|
| **XYZ2DXF** | *damfn.pic* | *damfn.dxf* |
| | *fn.pic* | *fn.dxf* |
| | *halfdfn.pic* | *halfdfn.dxf* |
| | *shrink.pic* | *shrink.dxf* |

Run AutoCAD to view the XYZ3D pictures that were just converted into DXF files (Figures A.7 to A.9).

Note that similar graphs (Figures A.7 to A.9) can now be generated using AutoCAD Release 10 or higher. For example, *mesh3d.dxf* file and the AutoCAD "HIDE" command can be used to produce the hidden line graph shown in Figure A.7

76

## 2. STATIC ANALYSIS RUNS

### A. GDAP Runs:

The input file for static run is **exstat.in**. This includes gravity and hydrostatic loads for the full reservoir condition. Gravity and hydrostatic loads are designated as load cases 1 and 2 and are applied separately. Nodal displacements and element stresses for each load case are calculated and stored in files *"statdsp"* and *"statstr"* for post-processing. To perform static analysis, at the DOS prompt type the following:

**> GDAP**

> Enter input file name: **exstat.in**
> Enter output file name: **exstat.out**

At the completion of static run, the following output files are generated.

| Program | Input Files | Output Files |
|---------|-------------|--------------|
| **GDAP** | **exstat.in** | *exstat.out* |
| | | *tape13.dat* |
| | | *tape8.dat* |
| | | *strcoord* |
| | | *statdsp* |
| | | *statstr* |

### B. Post-processing Runs:

Run POSTPRS to generate plot files for the computed nodal displacements and element stresses. The required input files for this run are shown in the following output files table. The processed displacements and stresses are stored separately in two different output files. The nodal displacements are saved in file *"disps.dxf"* where displacements for each load case are kept in separate AutoCAD LAYERs.

The static stresses for each load case are separated into the upstream and downstream stresses and are saved in files with extension *".str"*. Each stress file name starts with the letter *u* or *d* to indicate upstream or downstream face and ends with a number to show the load case. For example, *ustat2.str* indicates upstream static stresses for the load case 2. Each stress file contains arch, cantilever, shear, and principal stresses as well as other information needed to prepare stress contours and stress vector plots. To run POSTPRS, type the following:

**> postprs**

> Enter input file name: **poststat.in**
> Enter output file name: **poststat.out**

## OUTPUT FILES

| Program | Input Files | DXF Files | POSTPLT Files |
|---------|-------------|-----------|---------------|
| **POSTPRS** | *poststat.in* | *disps.dxf* | *ustat1.str* |
| | *tape13.dat* | | *dstat1.str* |
| | *tape8.dat* | | . |
| | *strcoord* | | . |
| | *statdsp* | | *ustatLL.str* |
| | *statstr* | | *dstatLL.str* |

### C. AutoCAD Run:

Run AutoCAD with "*disps.dxf*" as input to view the static deflections for each load case. The deflected shapes are drawn using the AutoCAD POLYLINE command. Use AutoCAD PEDIT command with spline curve-fitting option to smooth deflected shapes (Figure A.10).

### D. POSTPLT Runs:

Run the POSTPLT program to produce arch and cantilever stress contours, or to generate DXF vector plot files for principal stresses for the separate or combined static loads (1,2,...,LL load cases).

| Program | Input Files | Output |
|---------|-------------|--------|
| **POSTPLT** | *ustat1.str* | *stress-contours* |
| | *dstat1.str* | *vector-plots* |
| | ... | |
| | *ustatLL.str* | |
| | *dstatLL.str* | |

In this example the gravity and hydrostatic stresses were calculated separately, and were combined during the post-processing. The stress contours for the combined loads are shown in Figures A.11 and A.12a. POSTPLT stores the combined stresses in files "*ustat12.str*" and "*dstat12.str*", with "*12*" indicating that load cases 1 and 2 are being combined. The vector plots of principal stresses for water load alone and for gravity plus water loads are given in Figures A.12b to A.12e.

## 3. <u>ADDED MASS OF RESERVOIR WATER</u>

The added mass of incompressible reservoir water may be calculated using either the generalized Westergaard method or the FE procedure. In the Westergaard method, the added mass of water is calculated by simply setting IADMAS parameter to 1 (Record B.1, Section 6.0, p 21); no reservoir modeling is required. Whereas in the FE procedure, first an FE model of the reservoir water is developed, and then the added mass is computed by using the INCRES program.

### A. Pre-Processing Run:

Use the previously generated **incres.in** file as input, and run INCRES with a nonzero IPLOT option (see the input description for INCRES, Section 7.0, p 62 of the main text) to generate a 3-D plot for the reservoir model. Depending on the IPLOT selected (Record B, Section 7.0), DXF only or both XYZ and DXF drawings can be produced.

> **incres**

Enter input file name: **incres.in**
Enter output file name: **incres.out**

| Program | Input Files | Output Files |
|---------|-------------|--------------|
| INCRES | incres.in | *incres.xyz* *incres.dxf* |

### B. XYZ3D Runs:

Run XYZ3D to draw "*incres.xyz*", a 3-D picture of reservoir model with the hidden lines removed. As discussed earlier in Step 1.C, p 76, "p-7" can be set to 1 or 3 to save the XYZ3D picture in a binary or ASCII format.

### OUTPUT   FILES

| Program | Input Files | (p-7 = 1) | (p-7 = 3) |
|---------|-------------|-----------|-----------|
| XYZ3D | *incres.xyz* | *incres.plt* | *incres.pic* |

### C. XYZ2DXF Run:

First run XYZ2DXF to translate "*incres.pic*" to an AutoCAD DXF file. Then execute AutoCAD to view reservoir model, to add captions, or to perform other tasks (Figure A.14).

| Program | Input Files | DXF Files |
|---------|-------------|-----------|
| XYZ2DXF | *incres.pic* | *incres.dxf* |

79

## D. FE Added-Mass Calculation:

Set the IPLOT option in *"incres.in"* to zero to compute the added mass of the reservoir water. The output files for this run include the standard output file, *incres.out*, and a binary added-mass file, *tape12.dat*. This binary added-mass file will be used as an input in the subsequent dynamic analyses.

> **incres**

Enter input file name: **incres.in**
Enter output file name: **incres.out**

| Program | Input Files | Output Files |
|---------|-------------|--------------|
| INCRES | incres.in | *incres.out* *tape12.dat* |

## 4. FREQUENCY AND MODE-SHAPE RUNS

### A. Flexible Foundation, Full Reservoir:

The lowest five vibration frequencies and mode shapes are calculated for the example dam model. The input files for this run are the standard input file, **exeigen.in**, and the added-mass file, *tape12.dat*, produced by running INCRES. Note that *tape12.dat* is not required when Westergaard method is used. The output files include the standard output, *exeigen.out*, and several restart tape files for the subsequent response analysis. In addition, mode shapes are stored in the binary file, *modshap* for post-processing.

> **> GDAP**
> Enter input file name: **exeigen.in**
> Enter output file name: **exeigen.out**

| Program | Input Files | Output Files |
|---------|-------------|--------------|
| **GDAP** | **exeigen.in** **tape12.dat** | *exeigen.out* *tape13.dat* *tape10.dat* *tape9.dat* *tape8.dat* *tape1.dat* *modshap* |

### B. Post-processing of Mode Shapes:

First run POSTPRS with posteigen.in as input to generate the AutoCAD DXF file *modes.dxf* which contains the mode shapes. Each mode shape is stored in a separate AutoCAD LAYER called *mode-1*, *mode-2*,... and so on. Now run AutoCAD to view the mode-shapes, to add captions, or to perform other graphics tasks (Figures A.15 to A.19). Initially the deflected shapes are drawn as straight line segments by using the AutoCAD POLYLINE command. If smoothing is desired, use AutoCAD PEDIT command with spline curve-fitting option to smooth the mode-shapes.

> **> postprs**
> Enter input file name: **posteigen.in**
> Enter output file name: **posteigen.out**

| Program | Input Files | DXF Files |
|---------|-------------|-----------|
| **POSTPRS** | **posteigen.in** *tape13.dat* *tape8.dat* *modshap* | *posteigen.out* *modes.dxf* |

## 5. RESPONSE-SPECTRUM RUNS

### A. Analysis Run:

The response-spectrum analysis is performed for three components of an example earthquake ground motion applied simultaneously in the stream, vertical, and cross-stream directions. The standard input file for this run is **exrspec.in**. It contains the earthquake response spectra shown in Figures A.20 and A.21 that were used as the seismic input. Other input files include the restart added-mass binary file, *tape12.dat*, and other files generated in the previous runs. Two output files are produced: the standard output file *exrspec.out*, and file *rmsstr* which contains SRSS stress values for the dam elements.

> **GDAP**
>> Enter input file name: **exrspec.in**
>> Enter output file name: **exrspec.out**

| Program | Input Files | Output Files |
|---------|-------------|--------------|
| GDAP | exrspec.in | exrspec.out |
| | tape12.dat | rmsstr |
| | tape10.dat | |
| | tape9.dat | |
| | tape8.dat | |
| | tape1.dat | |

### B. Post-processing Run:

Run POSTPRS with the input files specified in the following table to generate stress contour plot files. *Uspec1.str* and *dspec1.str* are stress files for the upstream and downstream faces of the dam which could be plotted separately or combined with the static stresses (see Step C , POSTPLT runs on following page).

> **postprs**
>> Enter input file name: **postspec.in**
>> Enter output file name: **postspec.out**

| Program | Input Files | POSTPLT Files |
|---------|-------------|---------------|
| POSTPRS | postspec.in | postspec.out |
| | tape13.dat | uspec1.str |
| | tape8.dat | dspec1.str |
| | strcoord | |
| | rmsstr | |

## C. POSTPLT Runs:

Run POSTPLT to produce contour plots for the arch and cantilever stresses. The stress contours can be drawn for the response spectrum alone or for the response-spectrum plus static stresses due to various static loading combinations computed previously.

> postplt

| Program | Input Files | Output |
|---------|-------------|--------|
| POSTPLT | uspec1.str dspec1.str | contour plots |

The contour plots of the envelope arch and cantilever seismic stresses computed based on the Westergaard added mass are shown in Figures A.22b and A.23b.

The seismic stress contours computed using the FE added mass are shown in Figures A.22a and A.23a. The total stresses for the combined seismic plus static loads (usta12.str and dsta12.str) are presented in Figures A.24 and A.25.

## 6. TIME-HISTORY RUNS

### A. GDAP Run:

Time-history analysis of the dam model is performed for three components of a specified earthquake ground motion shown in Figures A.26 to A.28. The ground acceleration time-histories are applied simultaneously in the stream, vertical, and cross-stream directions. The input files include the standard input *exthist.in* and the same restart binary files used in the response-spectrum analysis. The output files are the standard output file, *exthist.out*, and several binary files containing the nodal displacements and element stresses.

> GDAP

      Enter input file name: **exthist.in**
      Enter output file name: **exthist.out**

| Program | Input Files | Output Files |
|---------|-------------|--------------|
| **GDAP** | **exthist.in** | *exthist.out ; standard output* |
| | *tape12.dat* | *thdisp1 ; maximum nodal displacements* |
| | *tape10.dat* | *thdisp2 ; time-histories of selected nodal displacements* |
| | *tape9.dat* | *thstr1 ; maximum arch, cantilever, and shear stresses* |
| | *tape8.dat* | *thstr2 ; time-histories of selected element stresses* |
| | *tape1.dat* | *thstpr1; maximum and minimum principal stresses* |
| | | *thstpr2 ; time-histories of selected principal stresses* |

### B. Post-processing Run:

The input files for post-processing of the time-history results consist of **postth.in** constructed according to the description of input data for POSTPRS program and the previously generated displacement and stress time-history files. In addition, when static plus dynamic stresses are required, the desired static stresses are also provided as input. These are the upstream and downstream stresses for a particular loading combination that have been generated in a previous static analysis.

The output files include the envelope tensile and compressive stresses, concurrent stresses at critical time-steps, and time-history of critical stresses and displacements. These files are listed in the following table and are described in Step C , POSTPLT runs on the following page.

      > postprs
            Enter input file name: **postth.in**
            Enter output file name: **postth.out**

| Program | Input Files | Output Files | POSTPLT Files |
|---|---|---|---|
| POSTPRS | postth.in<br>*thdisp1*<br>*thdisp2* | *postth.out* | *thdisp.plt*<br>*thstr.plt* |
| | *thstr1*<br>*thstr2* | | *uthistp.str*<br>*dthistp.str*<br>*uthistn.str*<br>*dthistn.str* |
| | *ustatL.str*<br>*dstatL.str* | | *uconcur1.str*<br>*dconcur1.str*<br>..........<br>*uconcurN.str*<br>*dconcurN.str* |

## C. POSTPLT Runs:

Run POSTPLT to draw stress contours for arch and cantilever stresses, vector plots for principal stress, or time-history plots for the selected displacements and stresses. POSTPLT is an interactive plotting package and prompts user for the input file name and other information. The files with extension ".*str*" are stress contour and vector plot files, and those with extension ".*plt*" are time-history files of the nodal displacements or element stresses.

| Program | Input Files | Output Plots |
|---|---|---|
| POSTPLT | *thdisp.plt*<br>*thstr.plt* | *Displacement Histories*<br>*Stress Histories* |
| | *uthistp.str*<br>*dthistp.str*<br>*uthistn.str*<br>*dthistn.str* | *U/S Envelope Tersile Stresses (u= u/s, p= positive, str=stress)*<br>*D/S Envelope Tensile Stresses (d= d/s, p= positive, str=stress)*<br>*U/S Envelope Comp. Stresses (u= u/s, n= negative, str=stress)*<br>*D/S Envelope Comp. Stresses (d= d/s, p=negative, str=stress)* |
| | *uconcur1.str*<br>*dconcur1.str*<br>...<br>*uconcurN.str*<br>*dconcurN.str* | *U/S Concurrent Stresses at T1 (u= u/s, 1 = T1, str=stress)*<br>*D/S Concurrent Stresses at T1 (d= d/s, 1 = T1, str=stress)*<br>...<br>*U/S Concurrent Stresses at Tn (u= u/s, N= Tn, str=stress)*<br>*D/S Concurrent Stresses at Tn (d= d/s, N= Tn, str=stress)* |
| | | *Vector Plots of Principal Stresses* |

In this example only the combined static plus dynamic stresses are presented. The cracking strength of the concrete is arbitrarily set to 350 psi (2.41 megapascals). The contours of the envelope tensile stresses are displayed in Figures A.29 and A.30. All critical stresses exceeding the cracking strength of the concrete are cantilever type. There are three critical time-steps corresponding to the maximum cantilever stresses. The contours of concurrent stresses at two critical time-steps are presented in Figures A.31 and A.32. The vector plots of envelope principal stresses are shown in Figures A.32b and A.32c. Time-history of nodal displacements for two selected nodes are shown in Figure A.33. Time-histories of all stresses exceeding the cracking strength of the concrete are displayed in Figures A.35 to A 38. In each figure, two stress histories are shown; one for a point with the maximum stres. and the other for its pair on the opposite face of the dam.
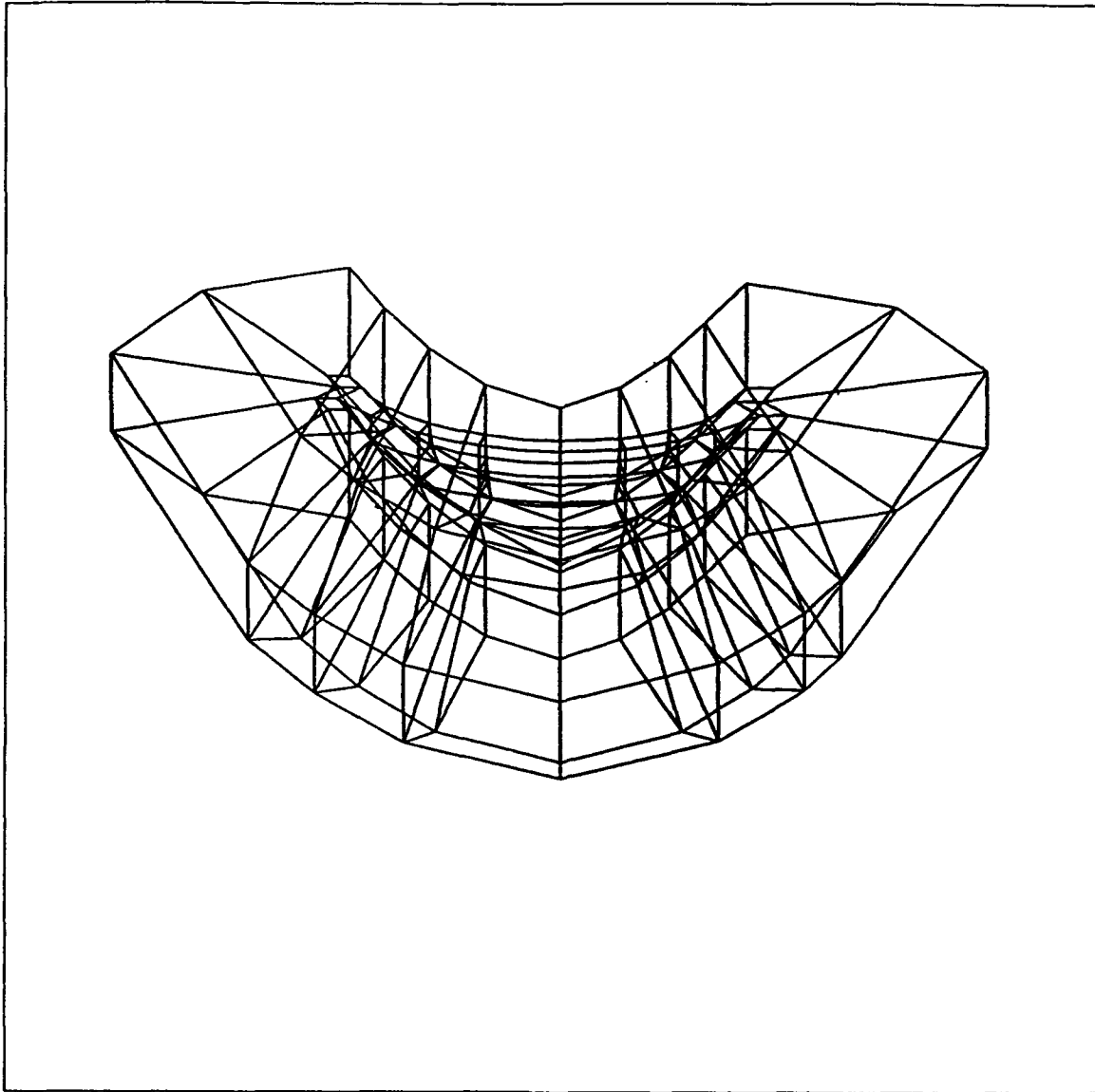
Figure A.1  Finite Element Dam-Foundation Model
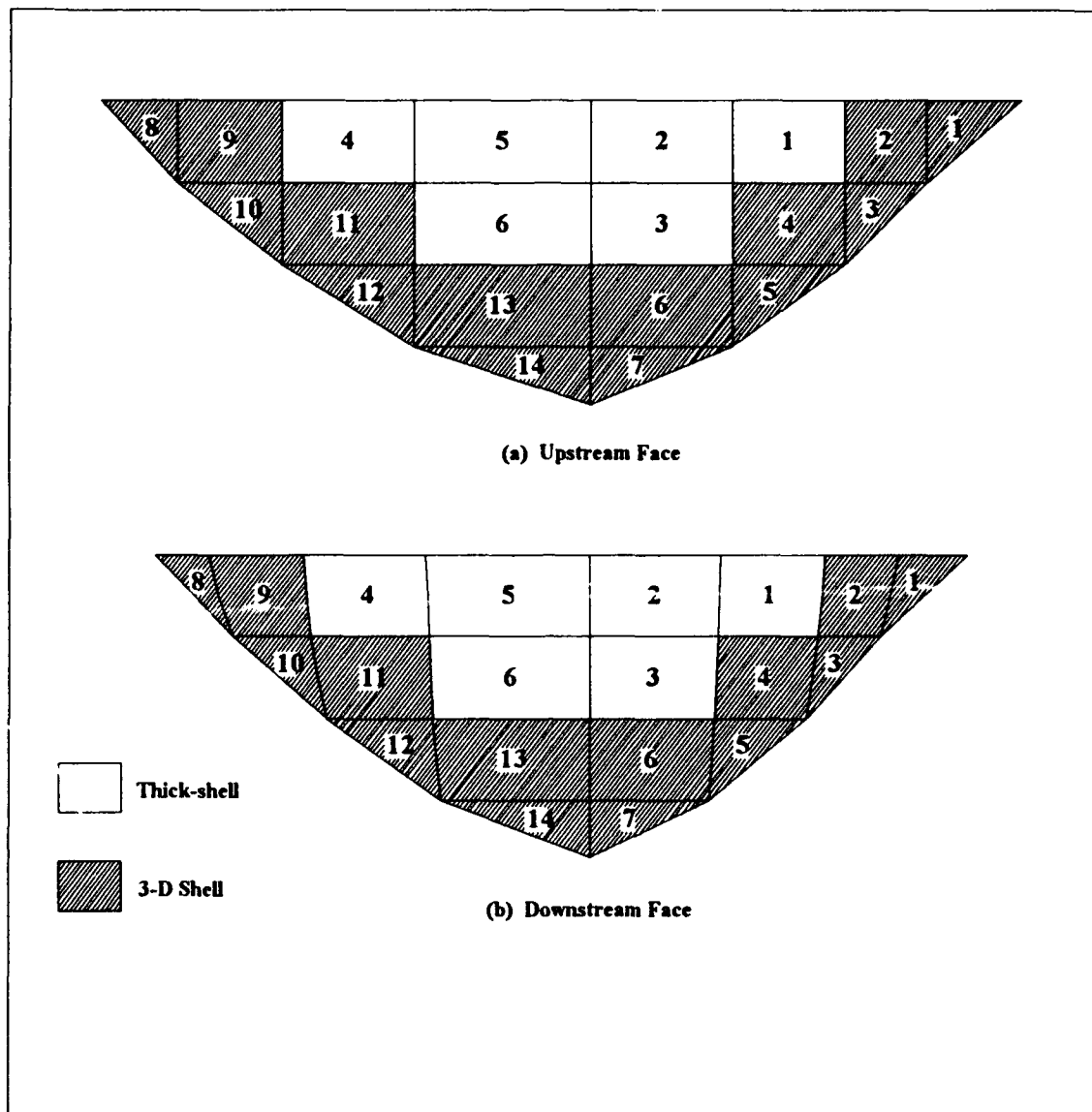
Figure A.2  Dam Element Numbers Shown on Upstream and Downstream Faces (Looking Downstream)
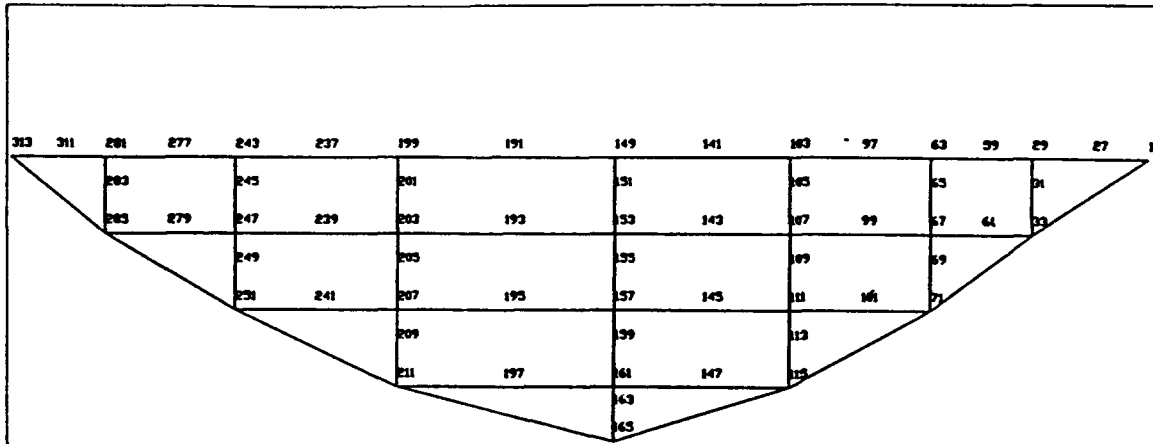
(a) Upstream Face
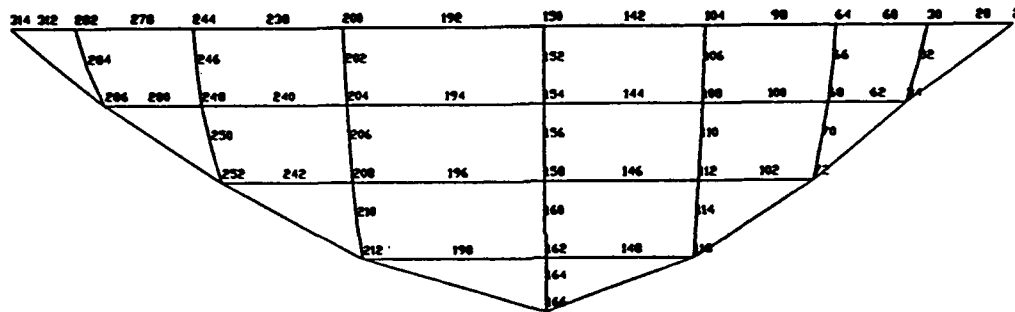
(b) Downstream Face

Figure A.3  Dam Node Numbers Shown on Upstream and Downstream Faces (Looking Downstream)

EL 350.00

EL 300.00

EL 250.00

EL 200.00
EL 177.00
EL 165.00

Crown
Section

Line of Centers
for Outer Segment
(Fillet Centers)

Line of Centers
for Center
Segment
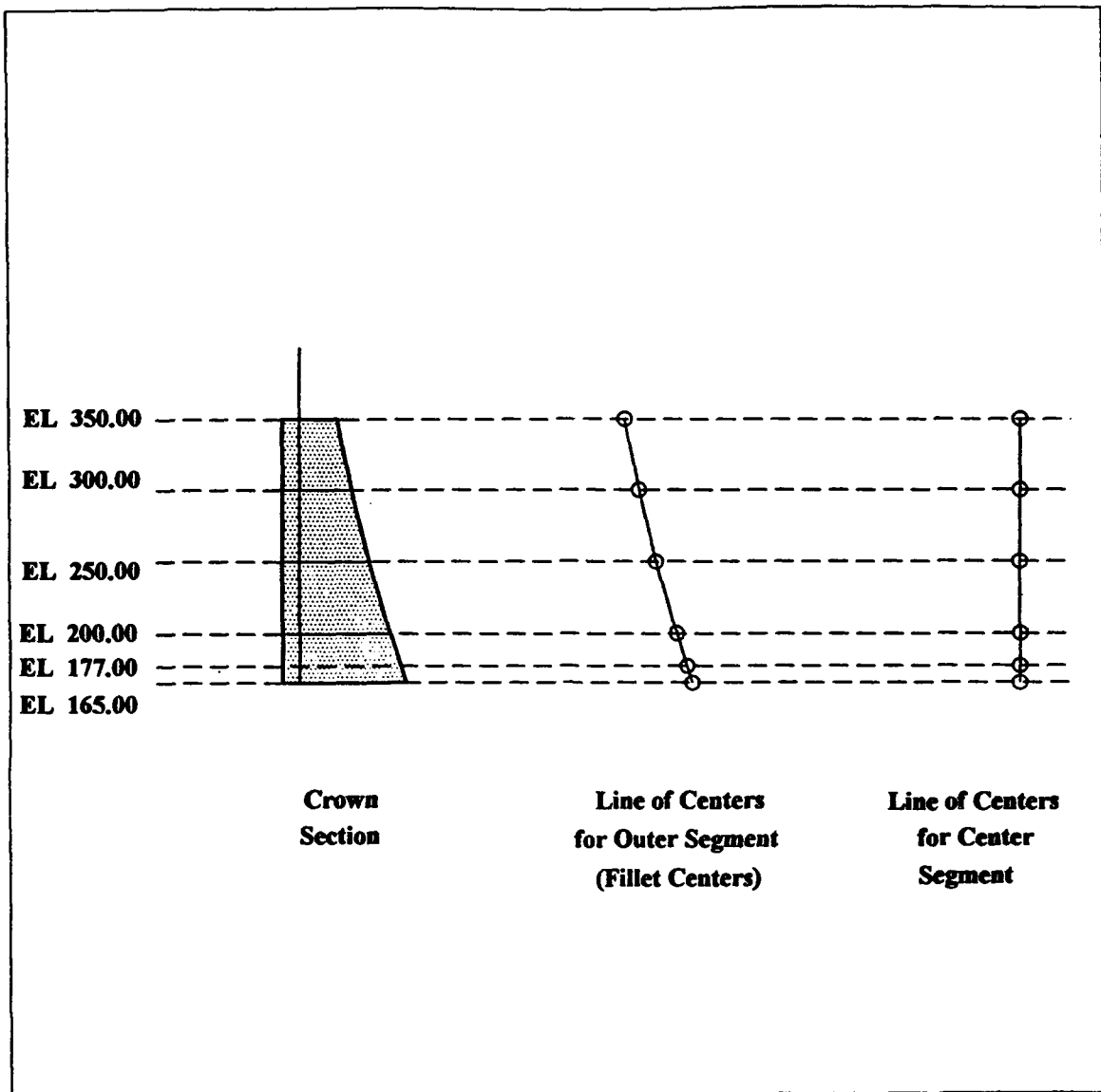
Figure A.4  Line of Centers for Example Dam Model
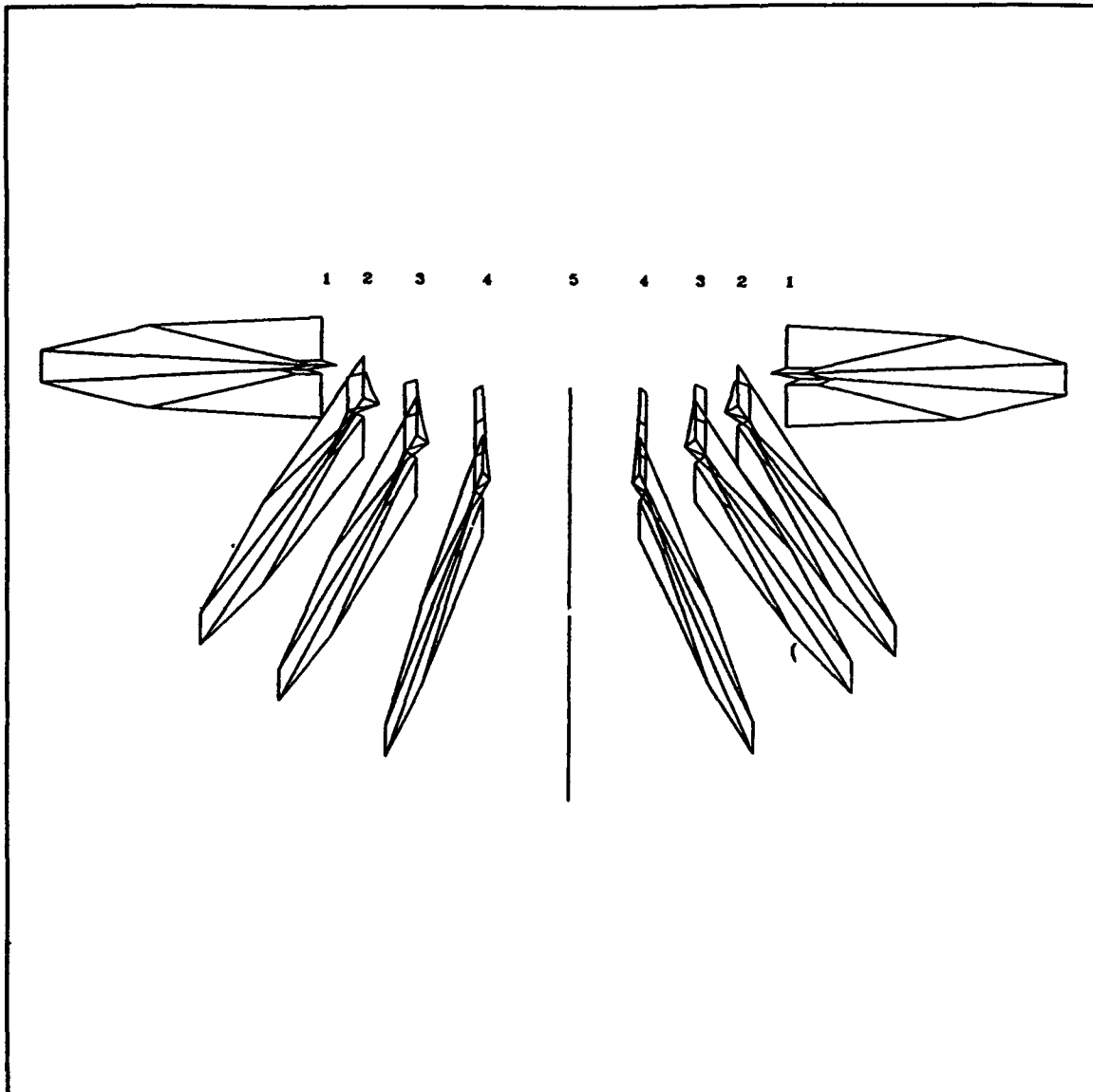
Figure A.5 Cantilever Sections

EL    350.00

EL    300.00

EL   ·250.00

EL    200.00
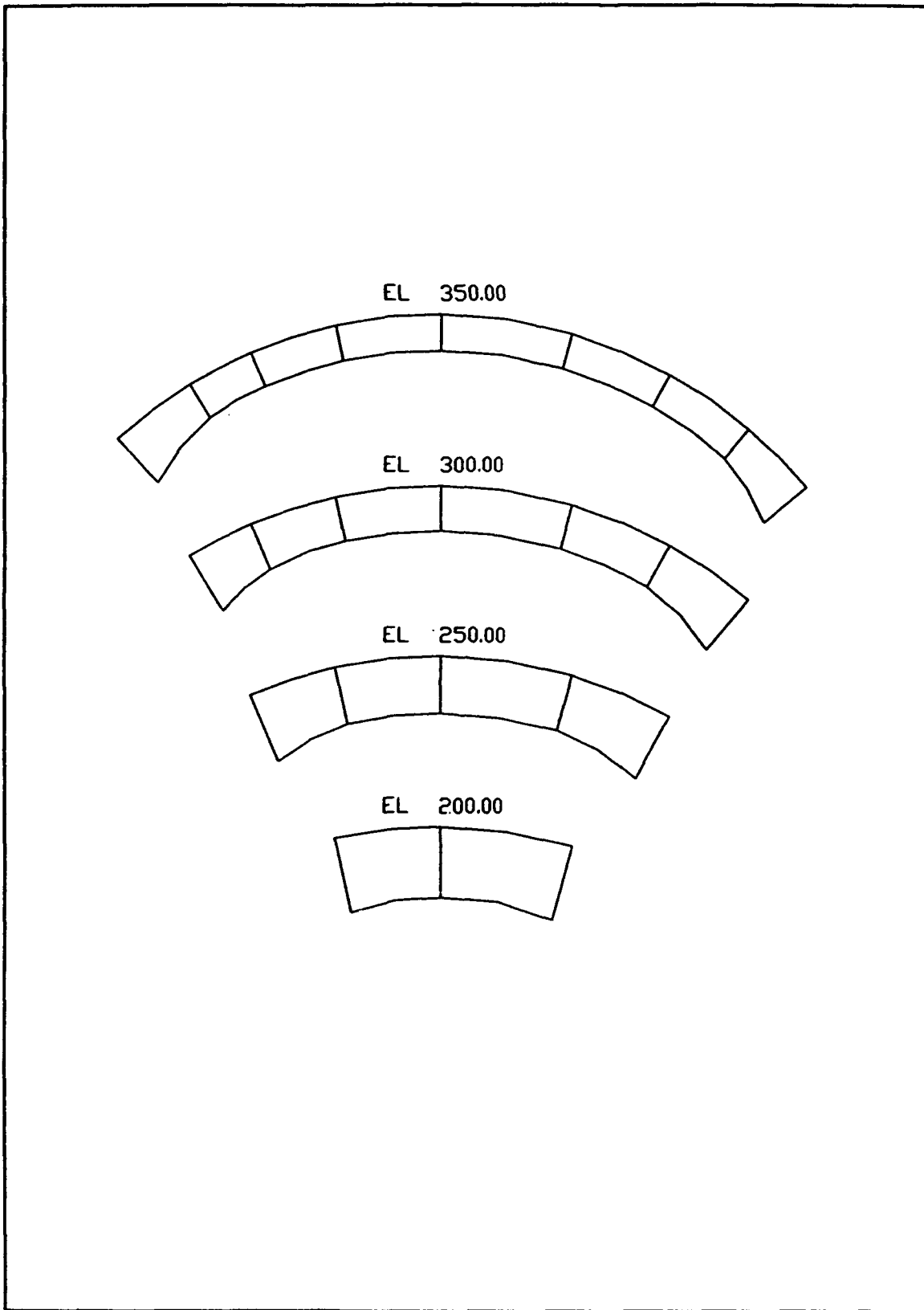
Figure A.6  Arch Sections at Mesh Elevations
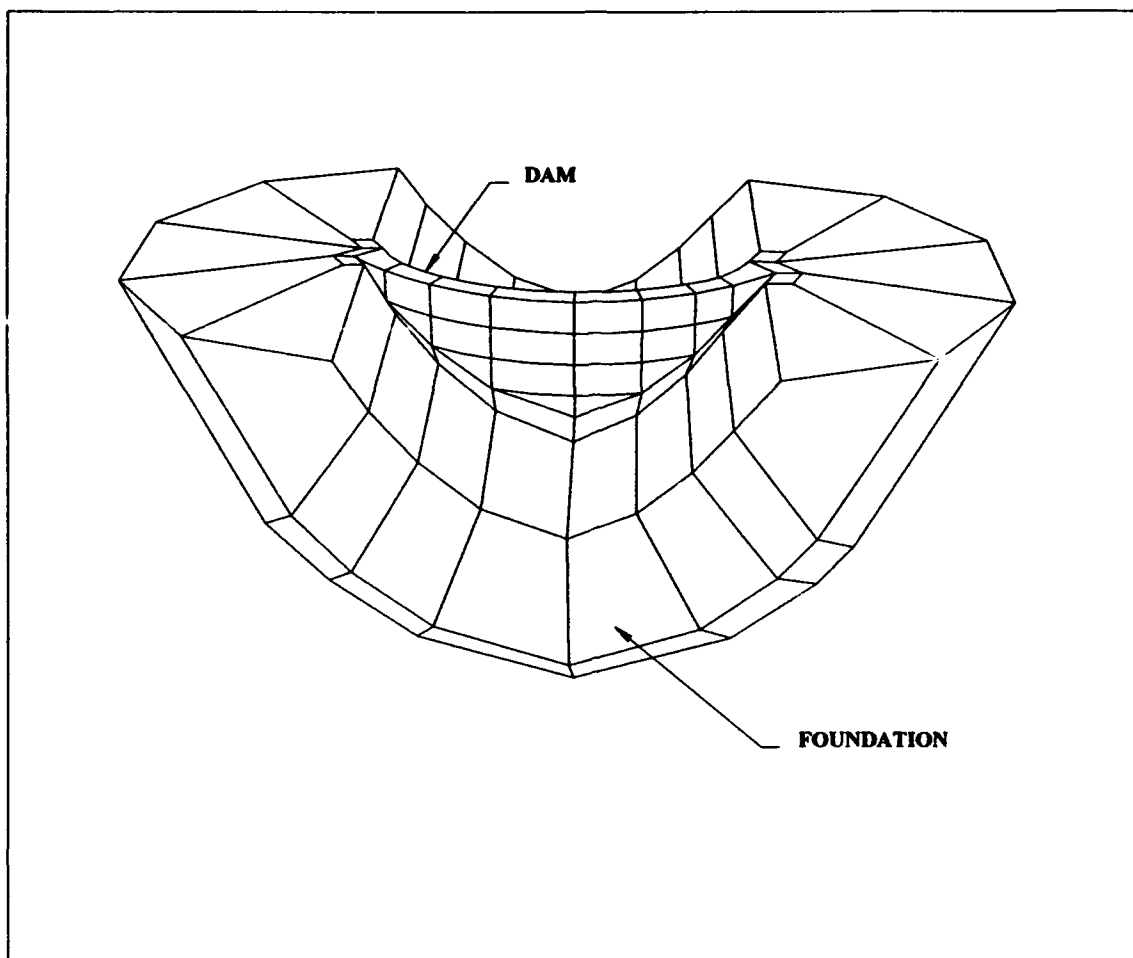
Figure A.7 Example Dam-Foundation Model (Hidden Lines Removed)

**FOOTPRINT OF DAM**
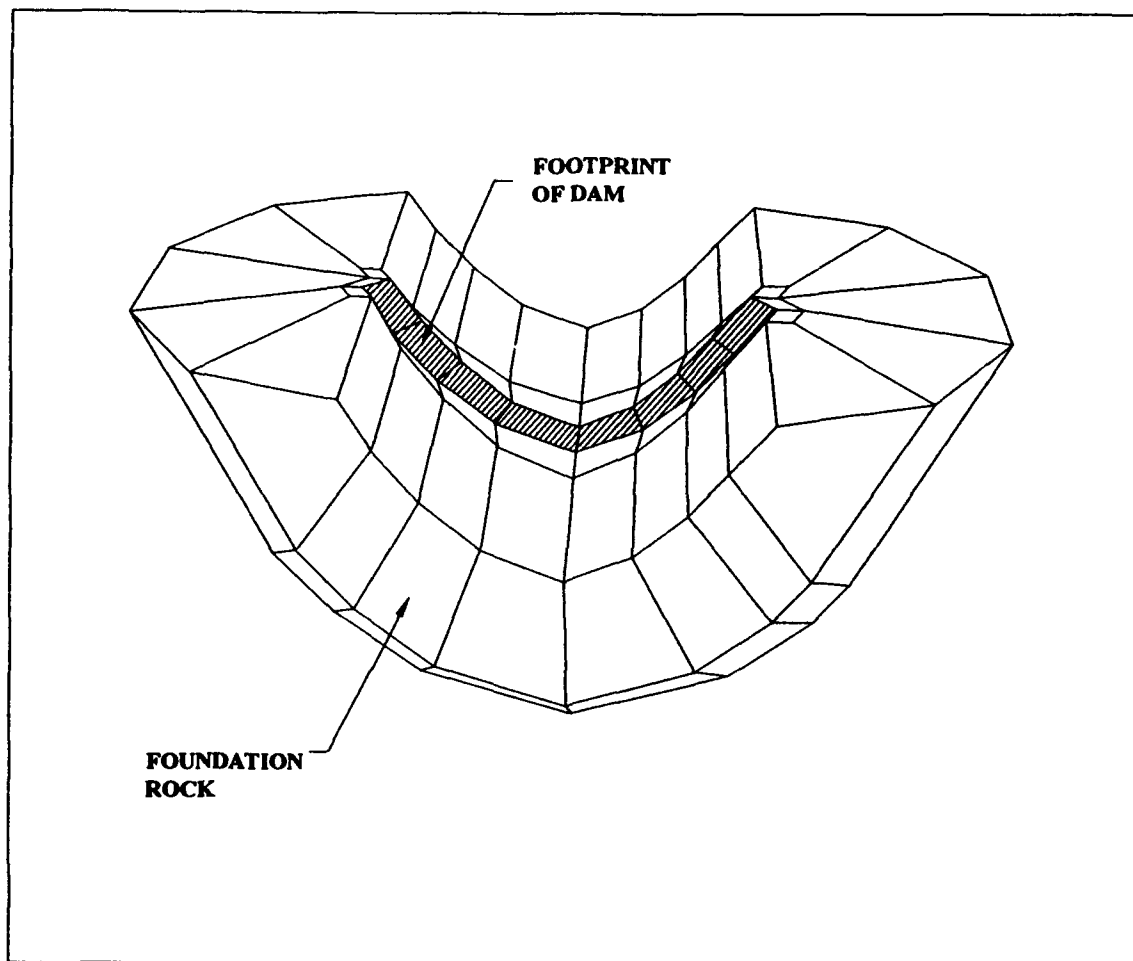
**FOUNDATION ROCK**

Figure A.8  Example Foundation Model (Type 1)
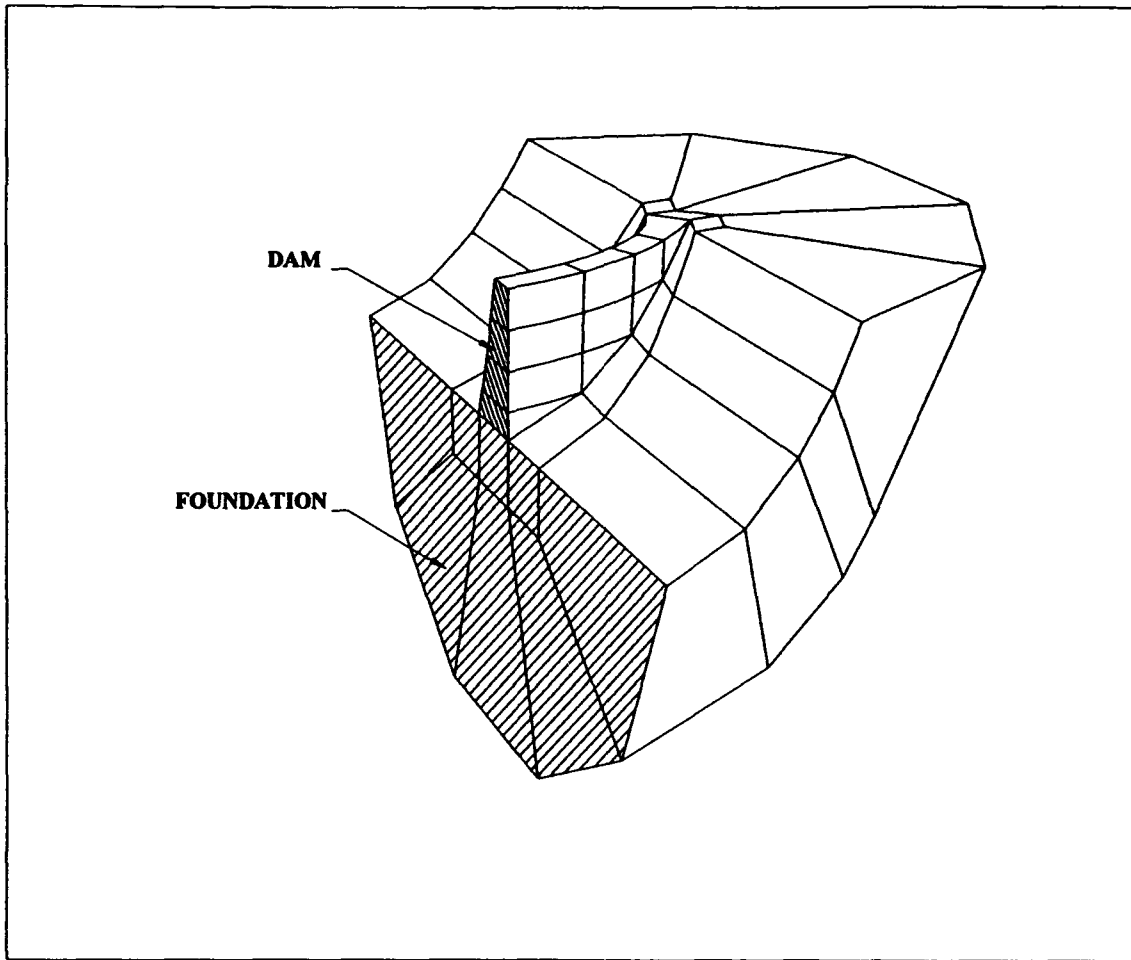
Figure A.9a  Example Dam Foundation Model (Right Portion)
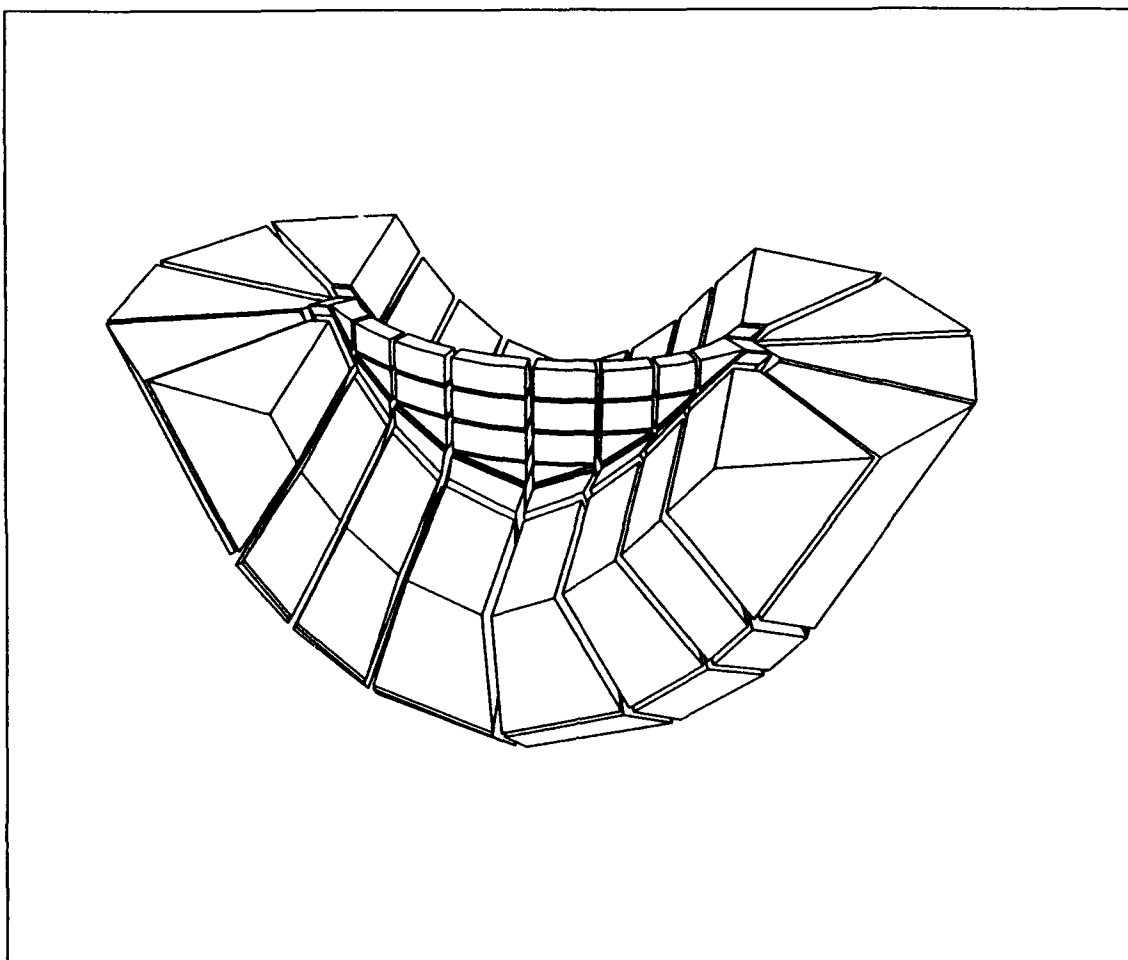
Figure A.9b  Shrink Plot of Dam and Foundation Models
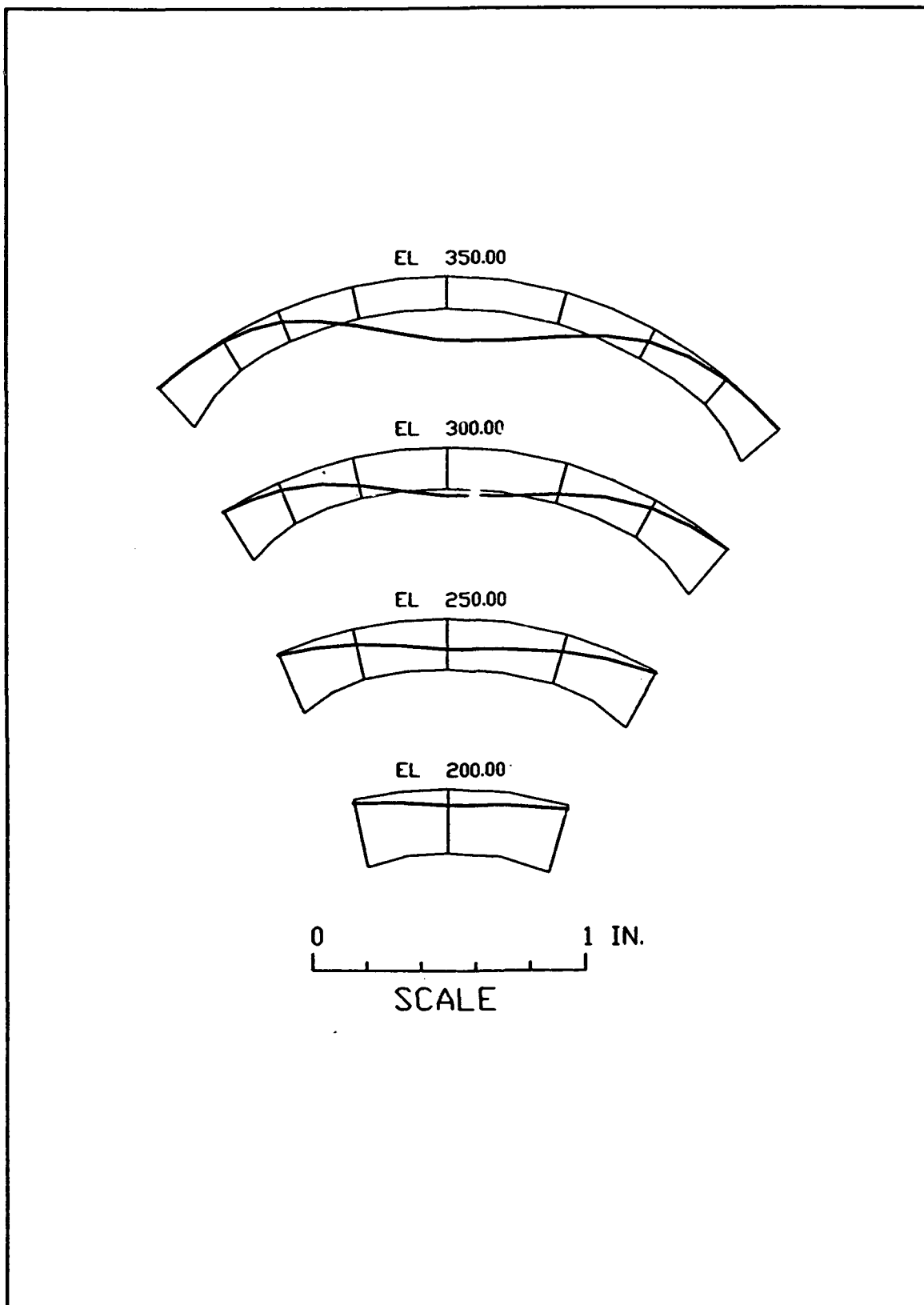
Figure A.10 Nodal Displacements Due to Hydrostatic Pressures (Water at El 350)
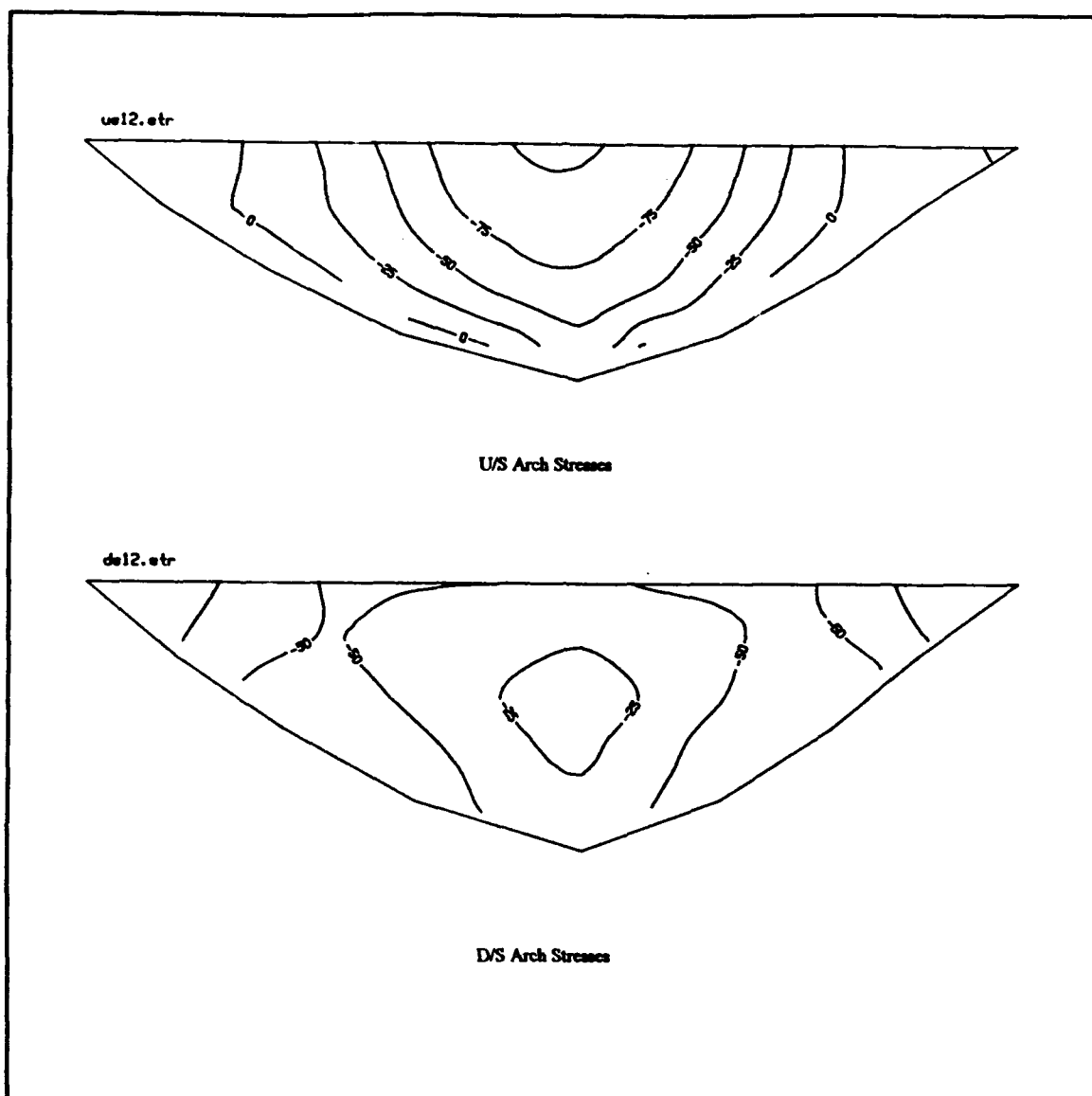
Figure A.11 Static Arch Stresses Due to Gravity + Hydrostatic Loads
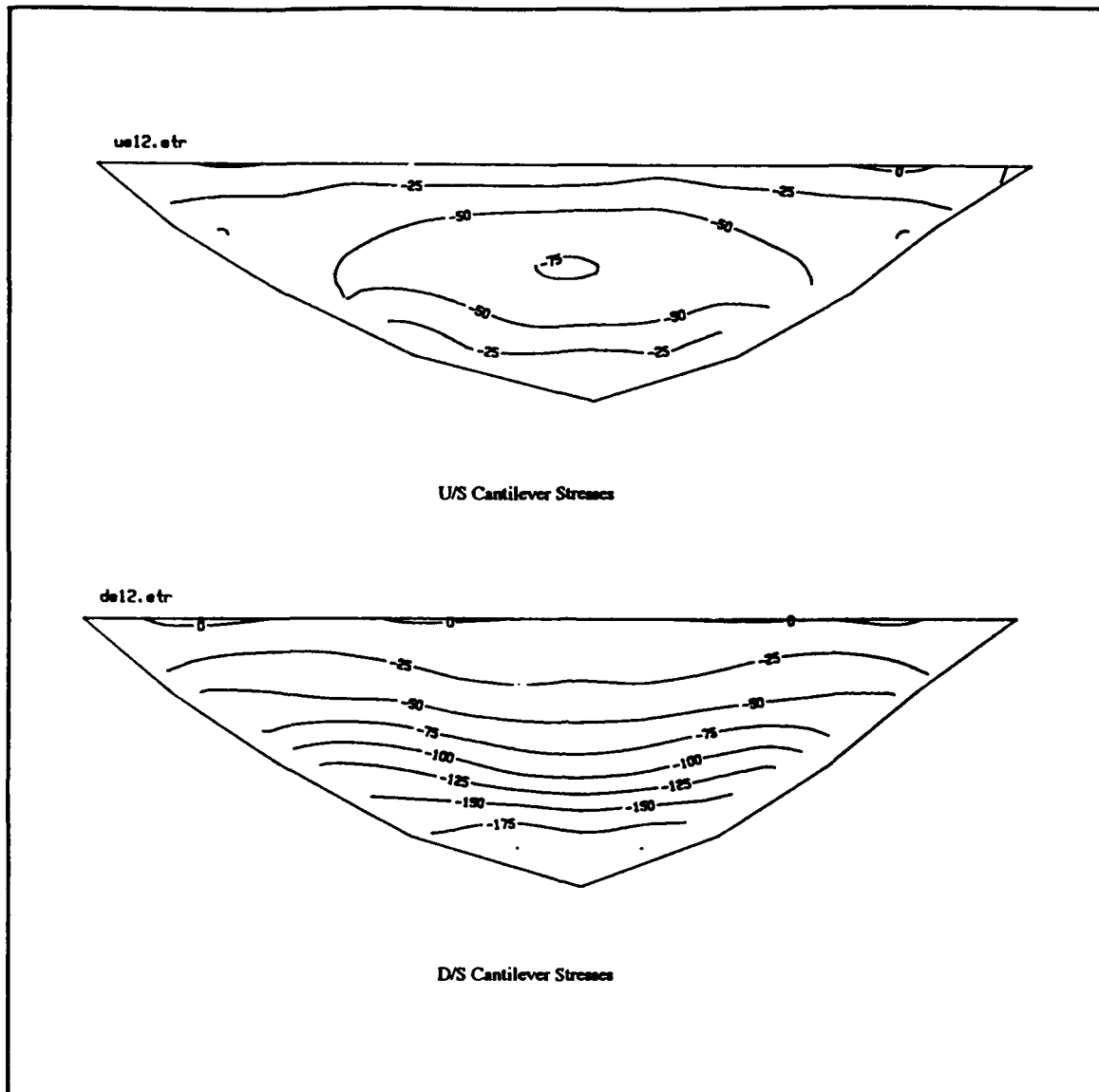
Figure A.12a  Static Cantilever Stresses Due to Gravity + Hydrostatic Loads

TENSION

COMPRESSION

0                    500 PSI

SCALE
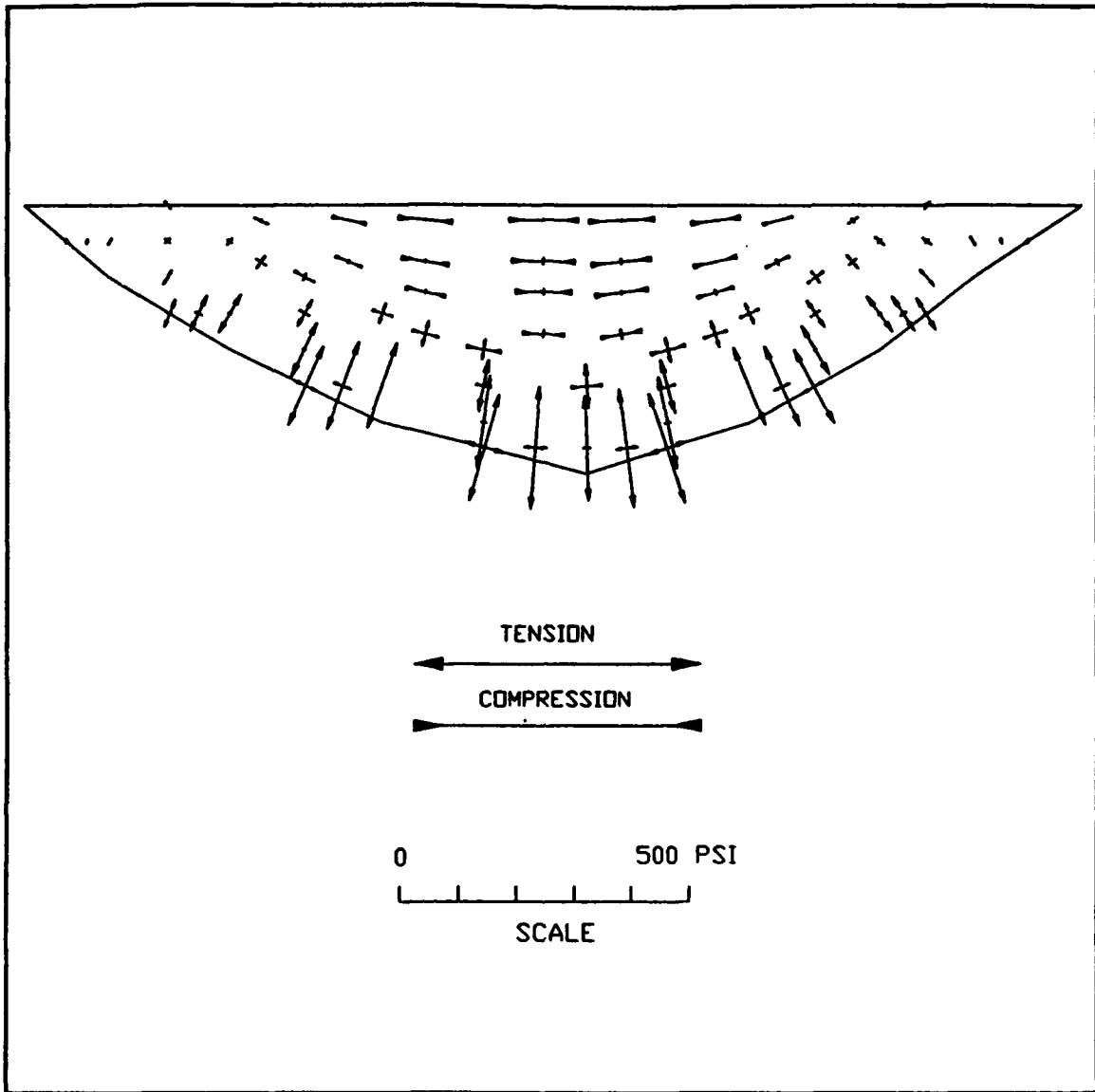
Figure A.12b  Static Principal Stresses (Water Loads)
Upstream Face

100

TENSION

COMPRESSION

0                    500 PSI

SCALE
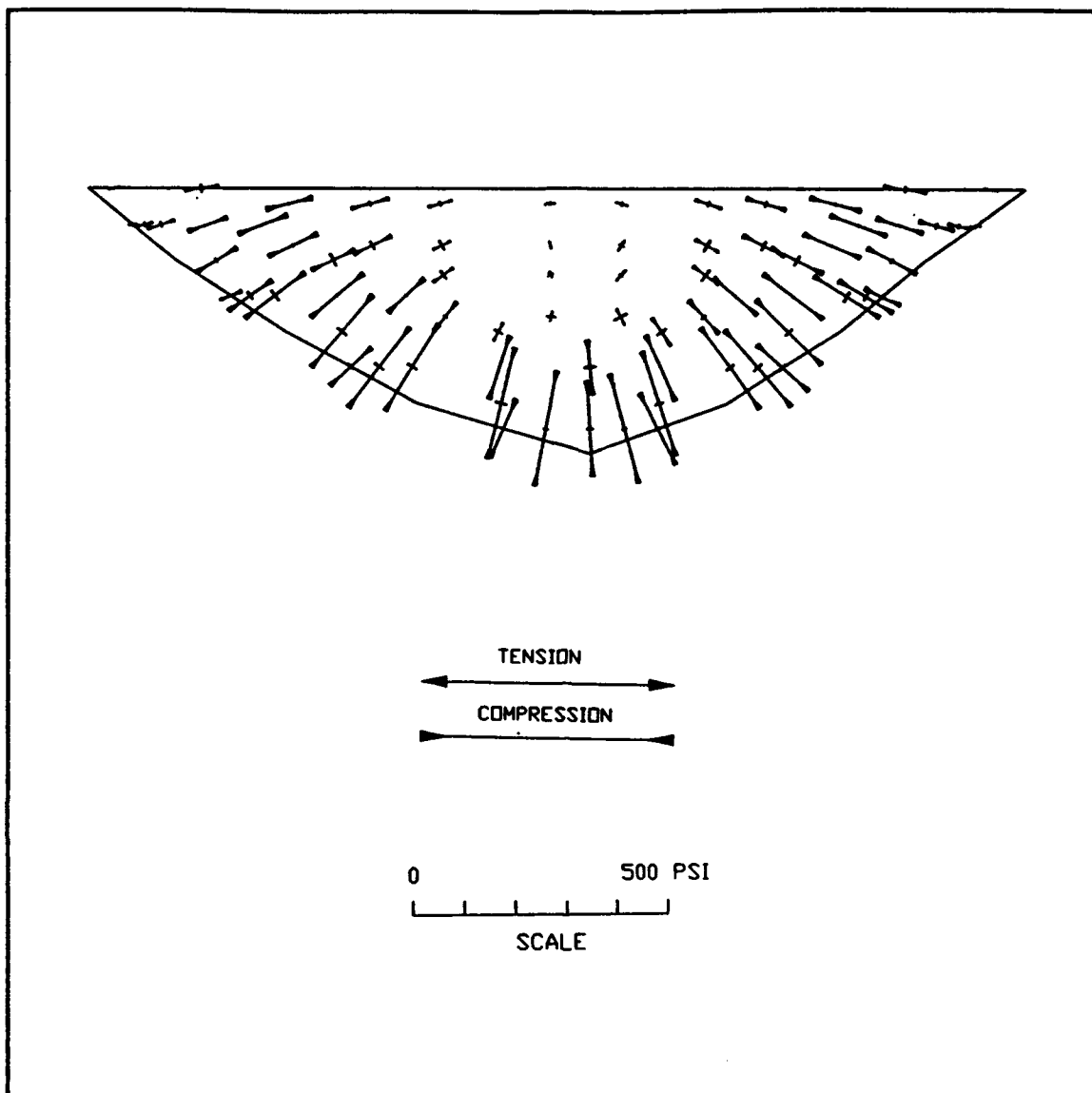
Figure A.12c  Static Principal Stresses (Water Loads)
Downstream Face

101
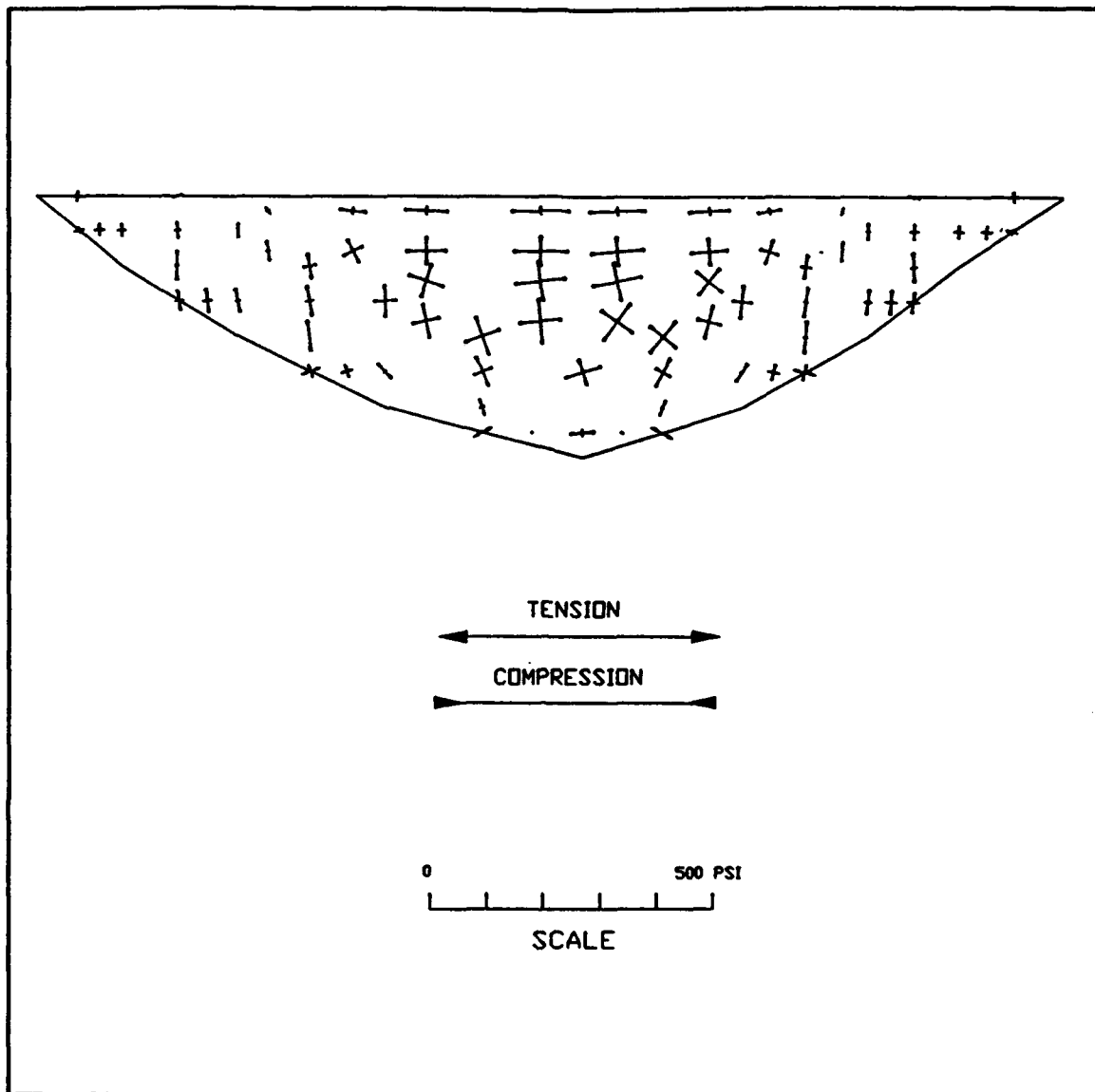
Figure A.12d  Static Principal Stresses (Gravity + Water)
Upstream Face
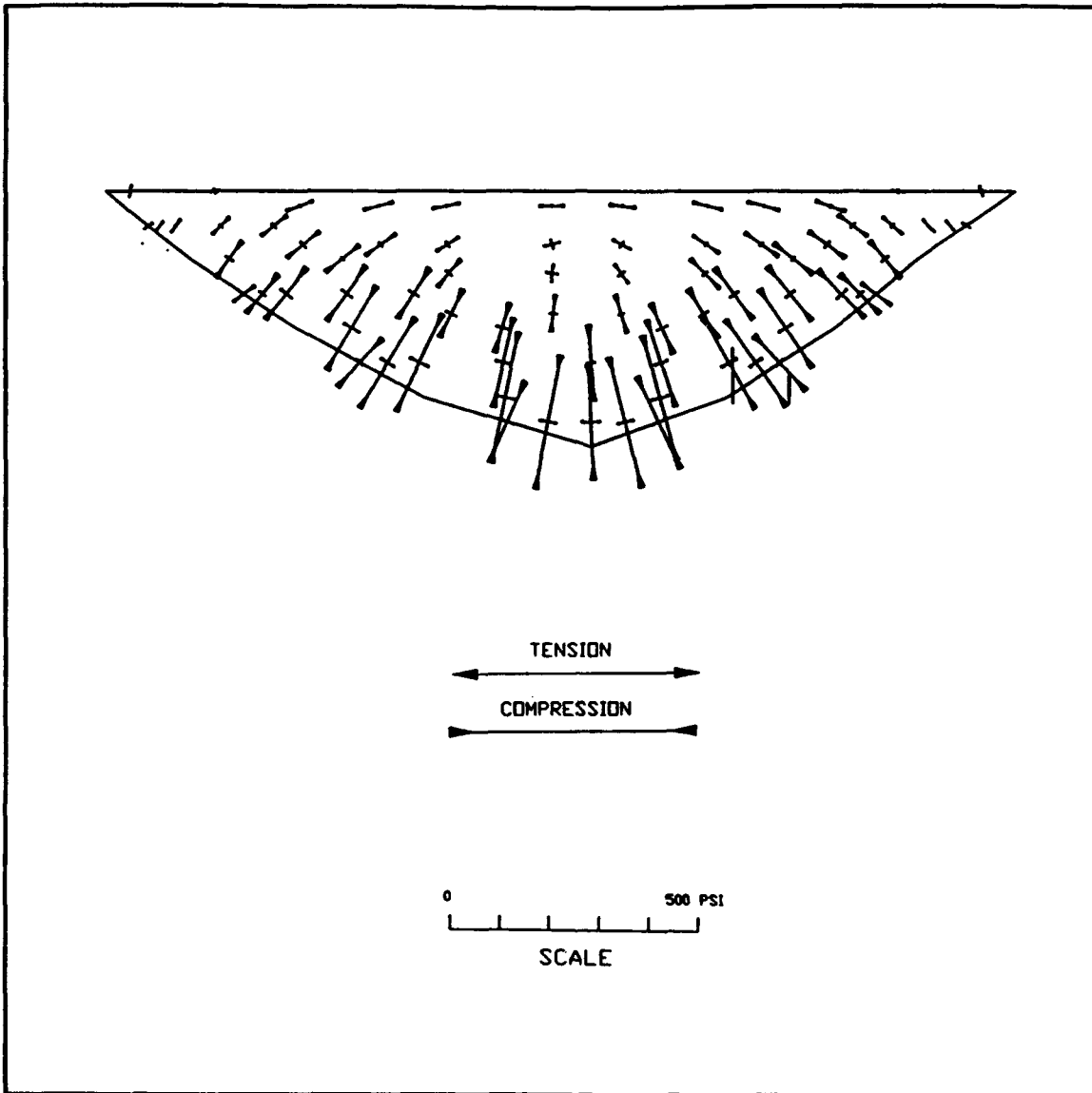
Figure A.12e  Static Principal Stresses (Gravity + Water)
Downstream Face
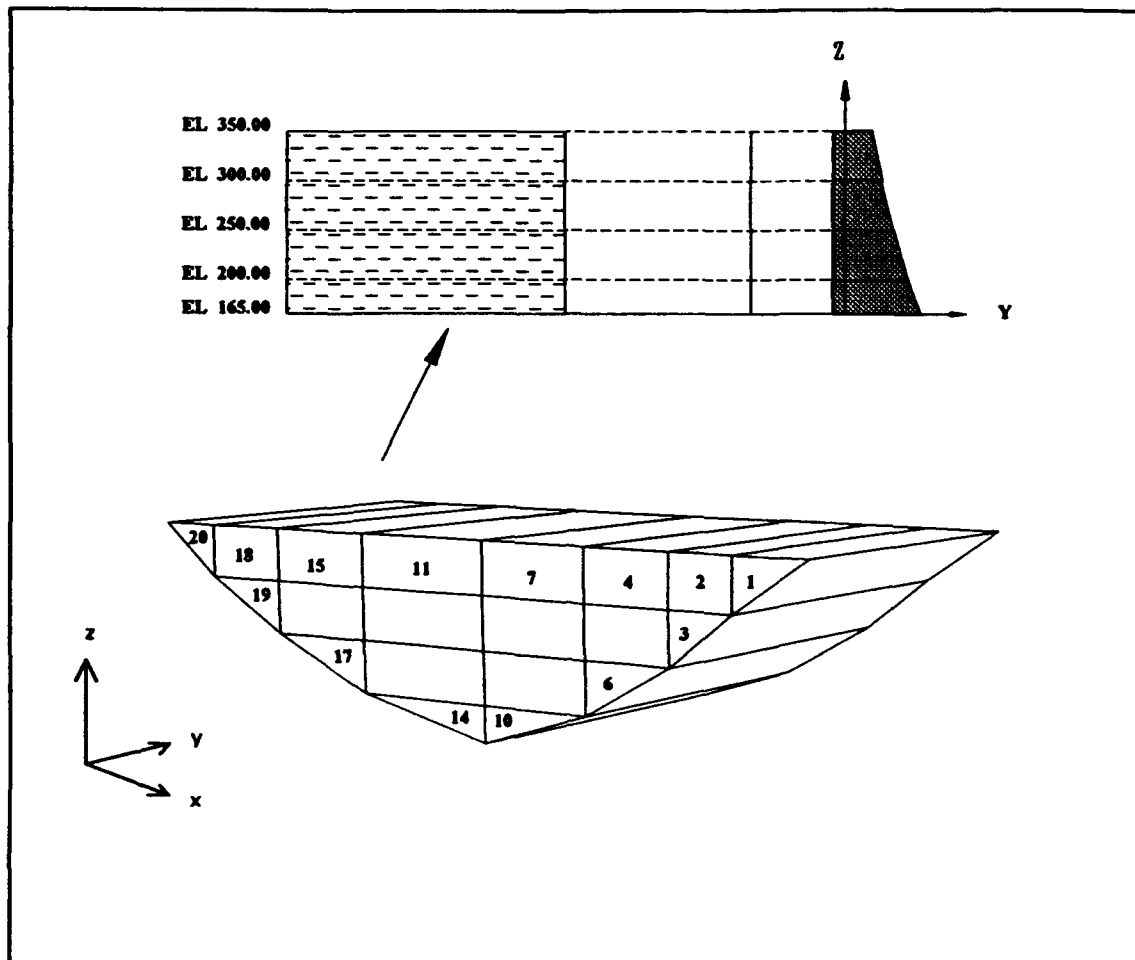
Figure A.13  Section View of Reservoir Water Model and Element Numbering Scheme
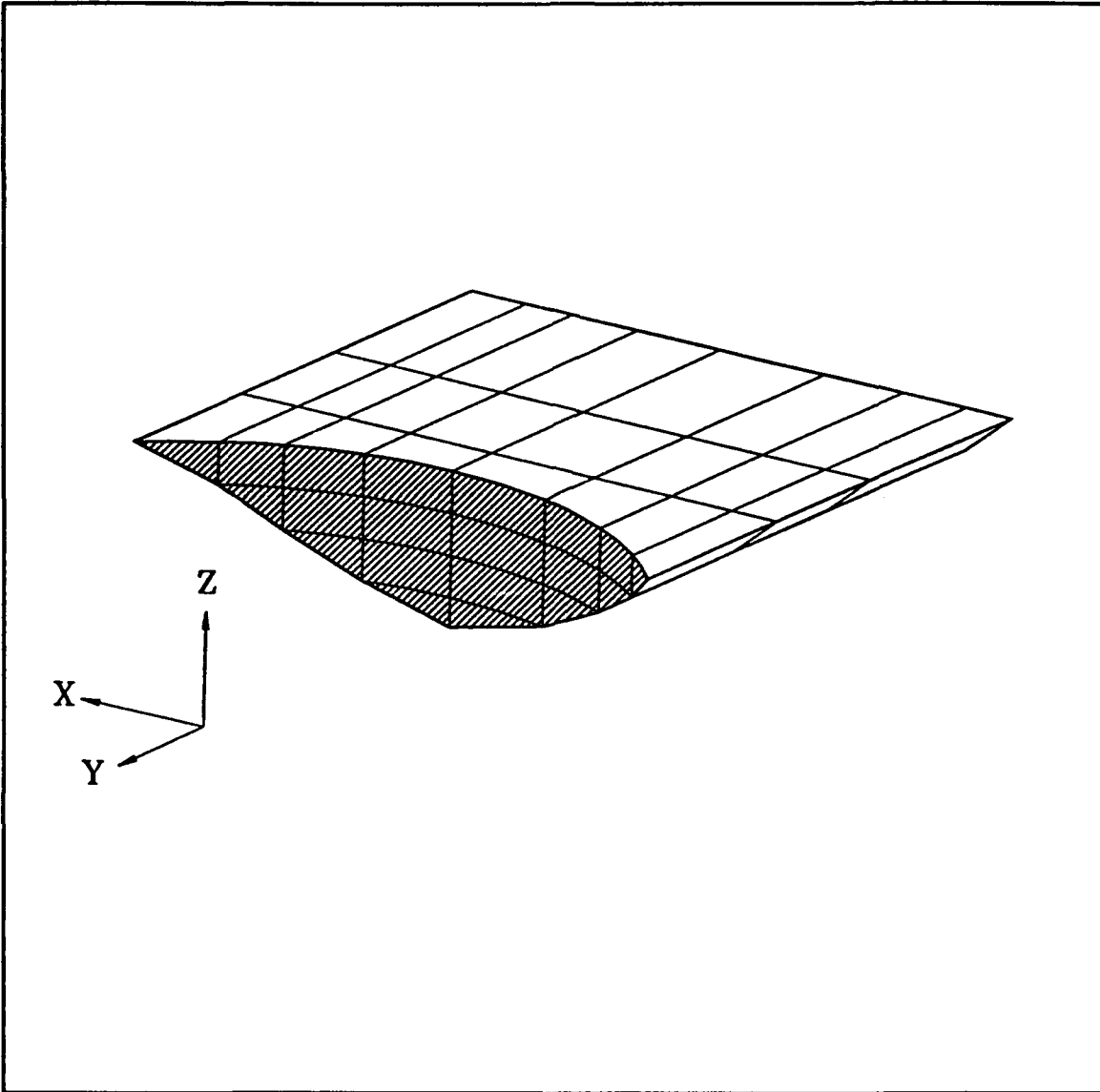
Figure A.14 Reservoir Water Finite Element Model

EL    350.00

EL    300.00

EL    250.00

EL    200.00

Figure A.15  Mode Shape 1

106

EL  350.00

EL  300.00

EL  250.00

EL  200.00

Figure A.16 Mode Shape 2

107

EL 350.00

EL 300.00

EL 250.00

EL 200.00
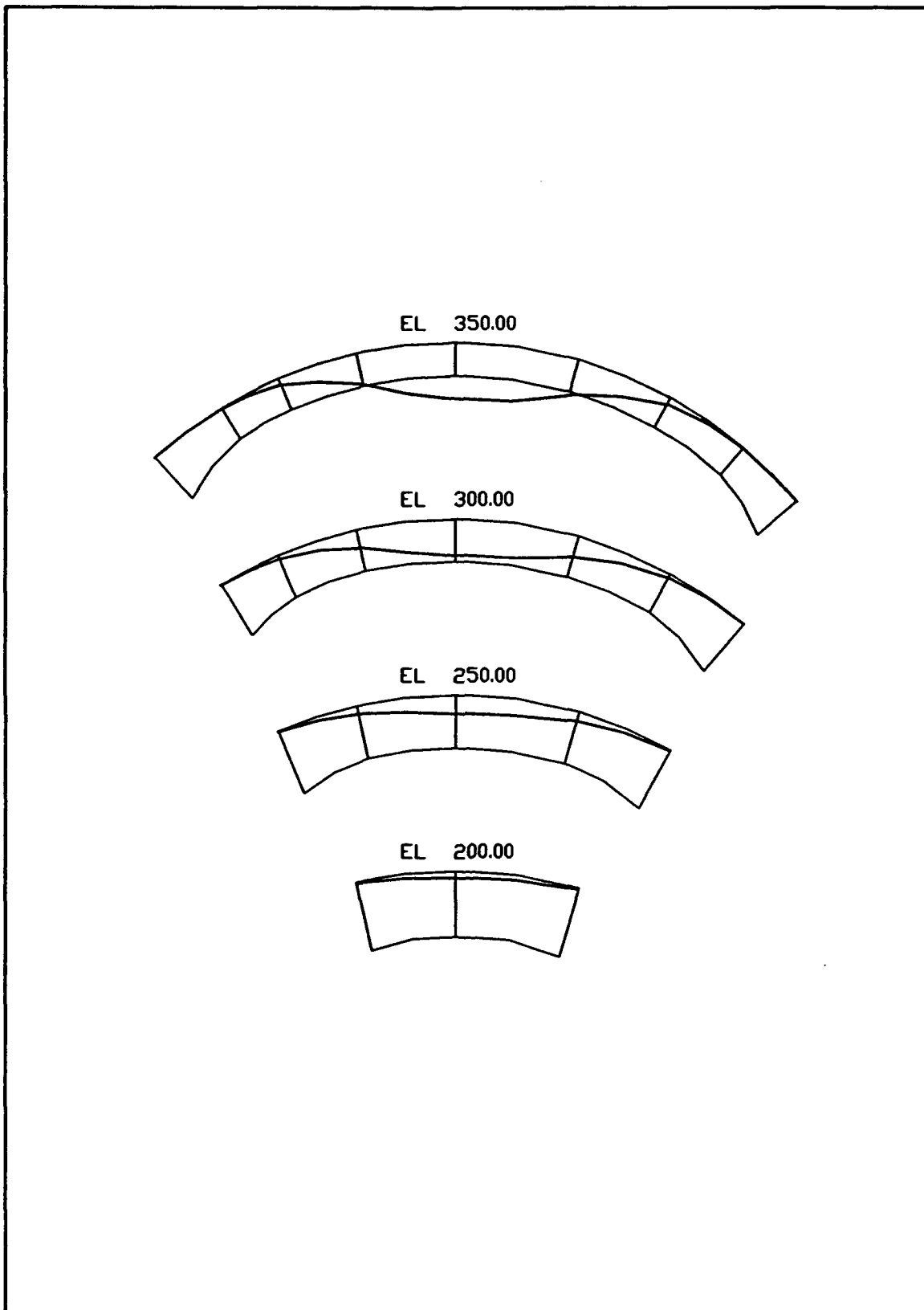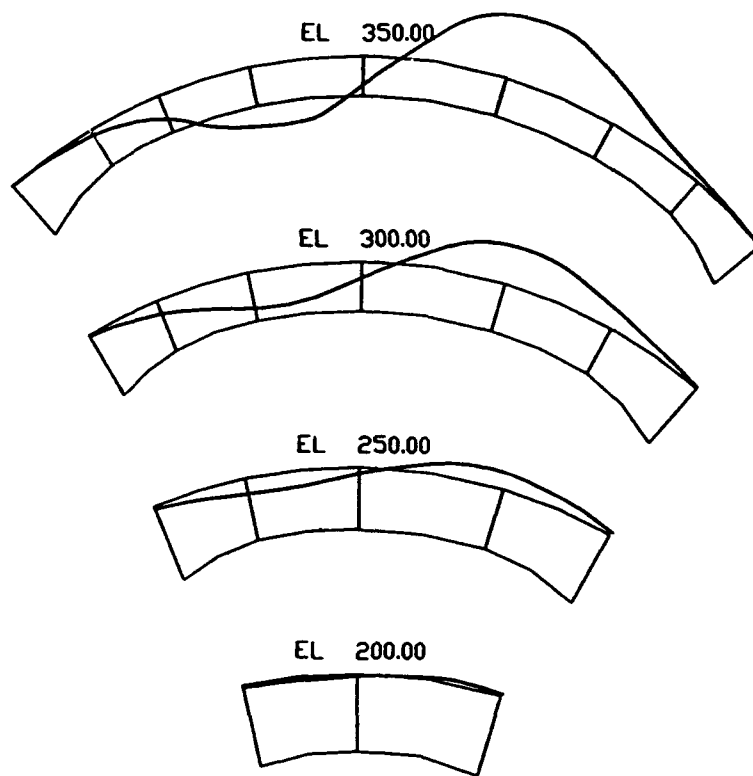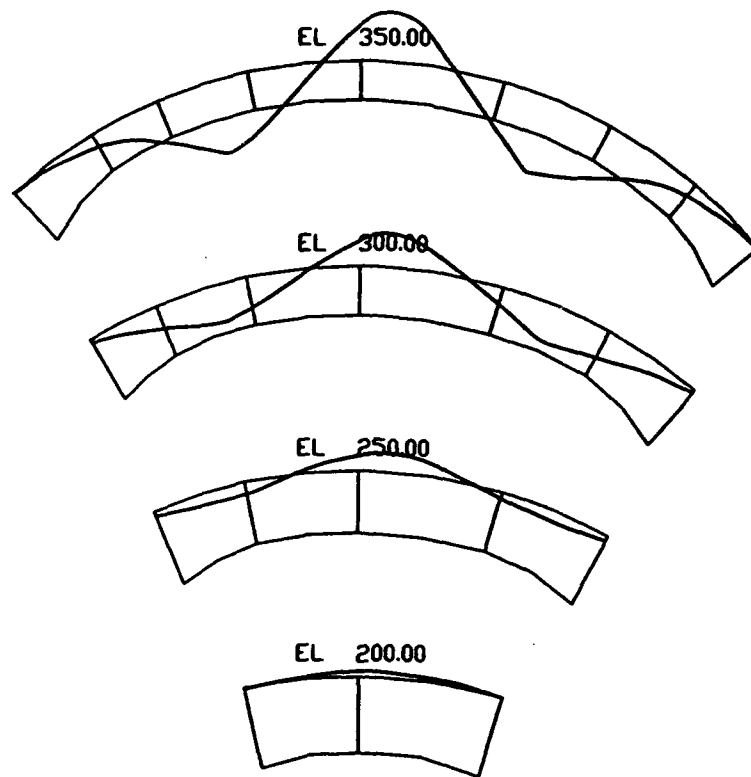
Figure A.17 Mode Shape 3

EL   350.00

EL   300.00

EL   250.00

EL   200.00

Figure A.18  Mode Shape 4

EL    350.00

EL    300.00

EL    250.00

EL    200.00
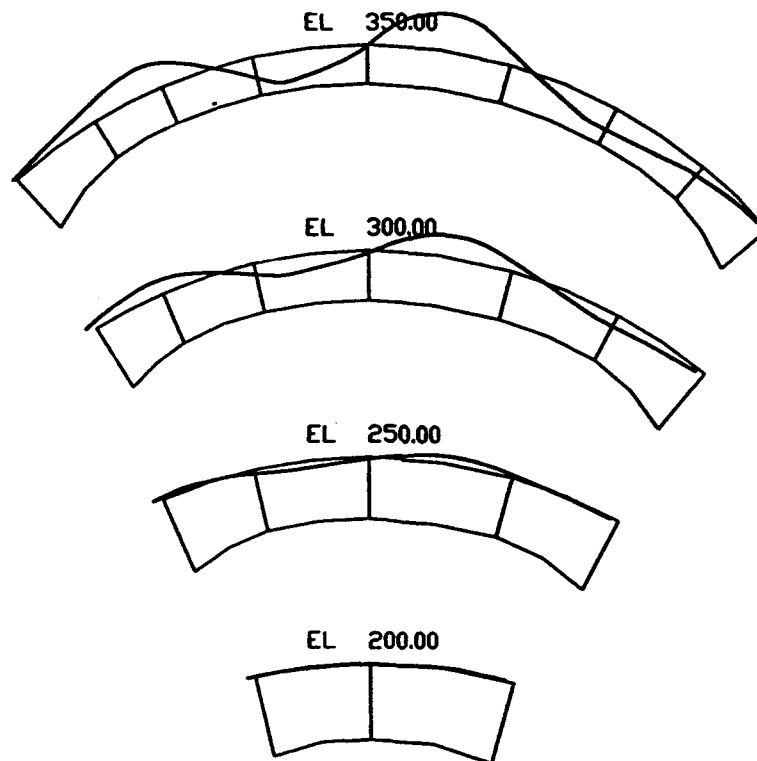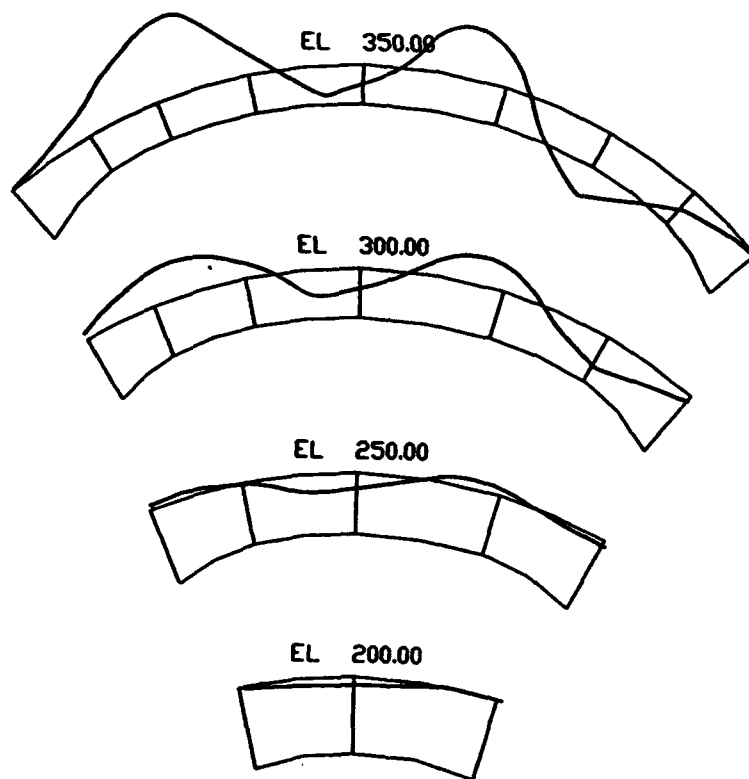
Figure A.19  Mode Shape 5

110
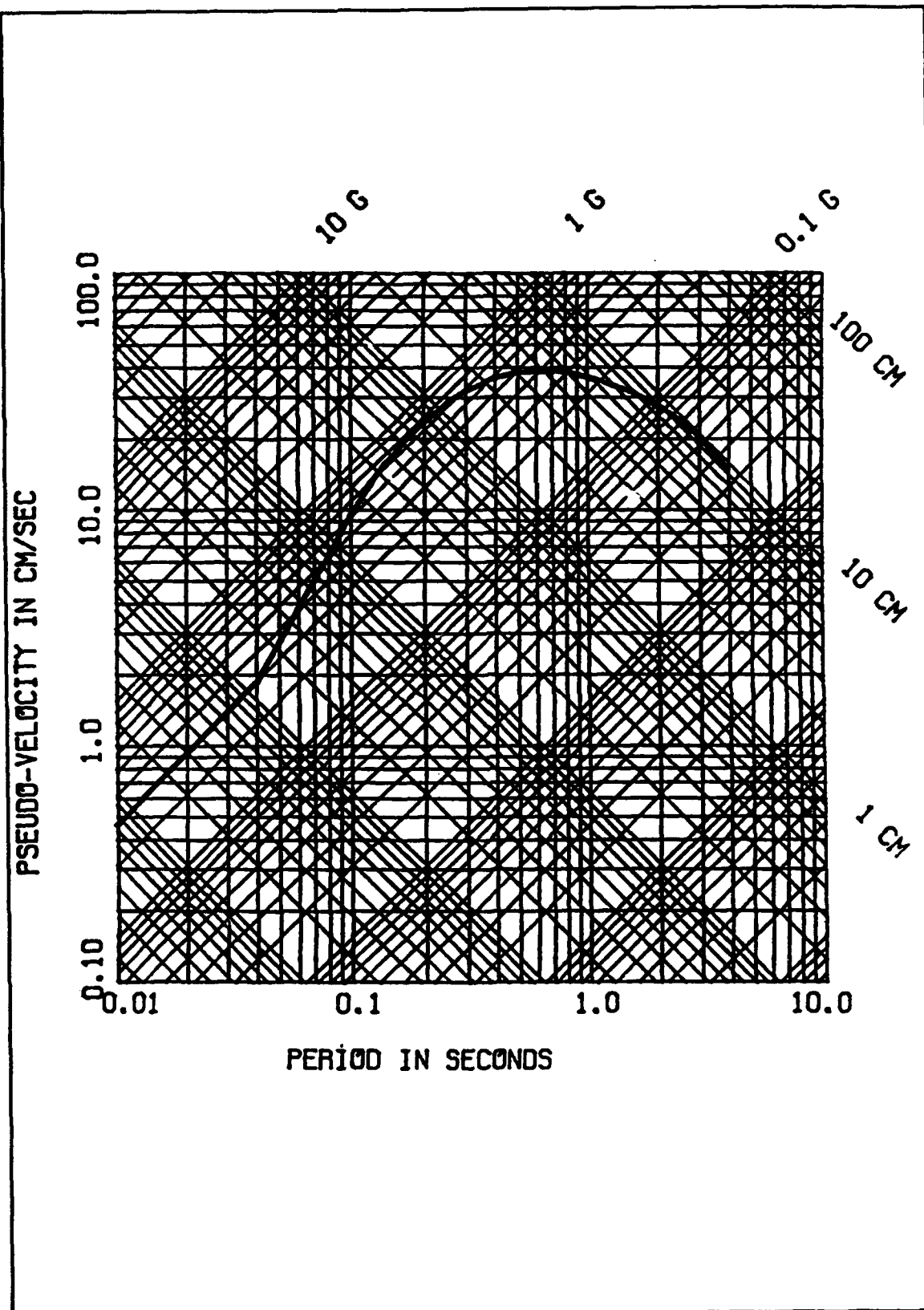
Figure A.20  Response Spectra of Horizontal Motions (5% Critical Damping)
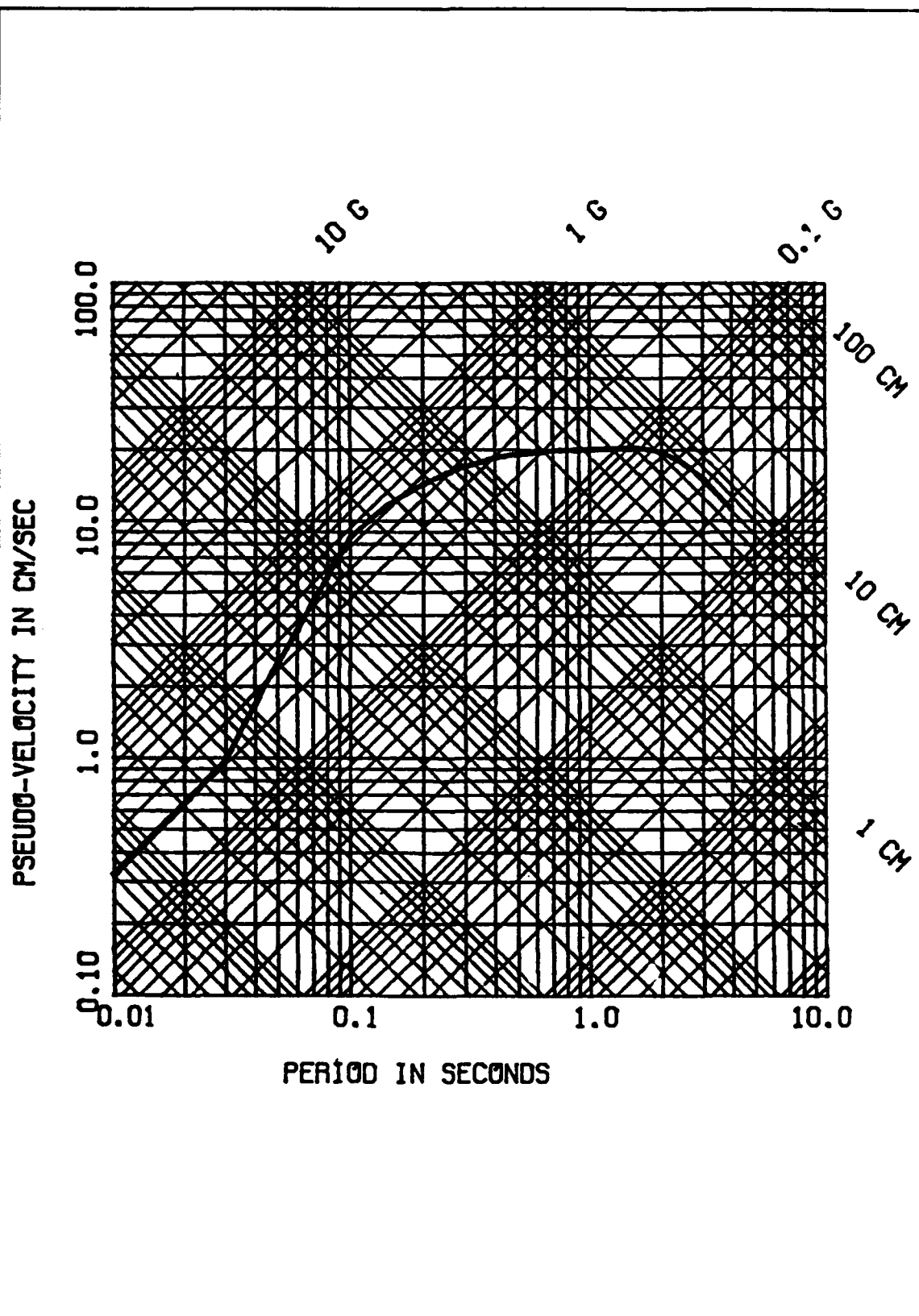
Figure A.21  Response Spectra of Vertical Motion (5% Critical Damping)

uspecl.str

U/S Arch Stresses

dspecl.str

D/S Arch Stresses
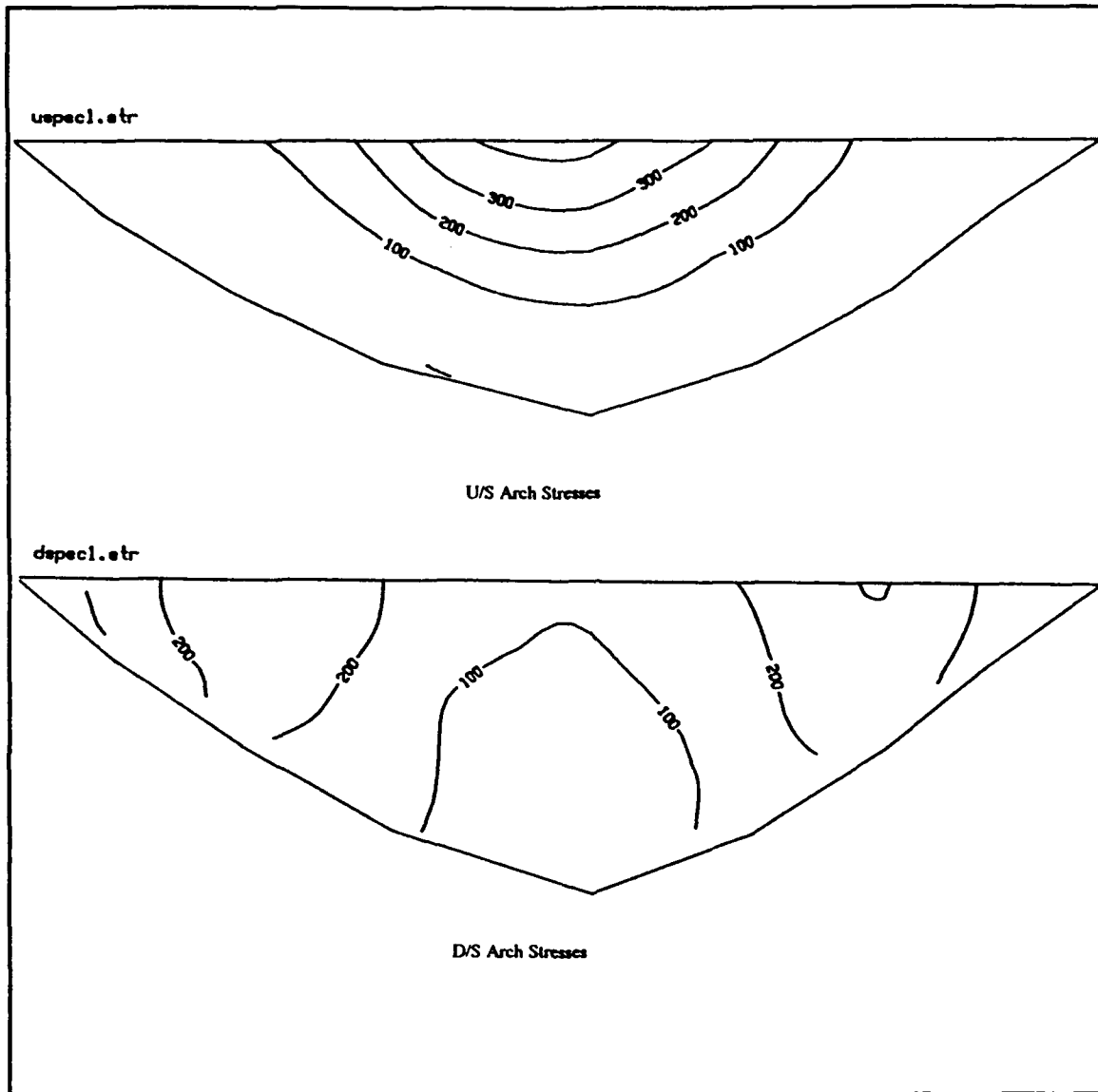
Figure A.22a  Dynamic  Stresses Due to Response Spectrum (FE Added Mass)

113
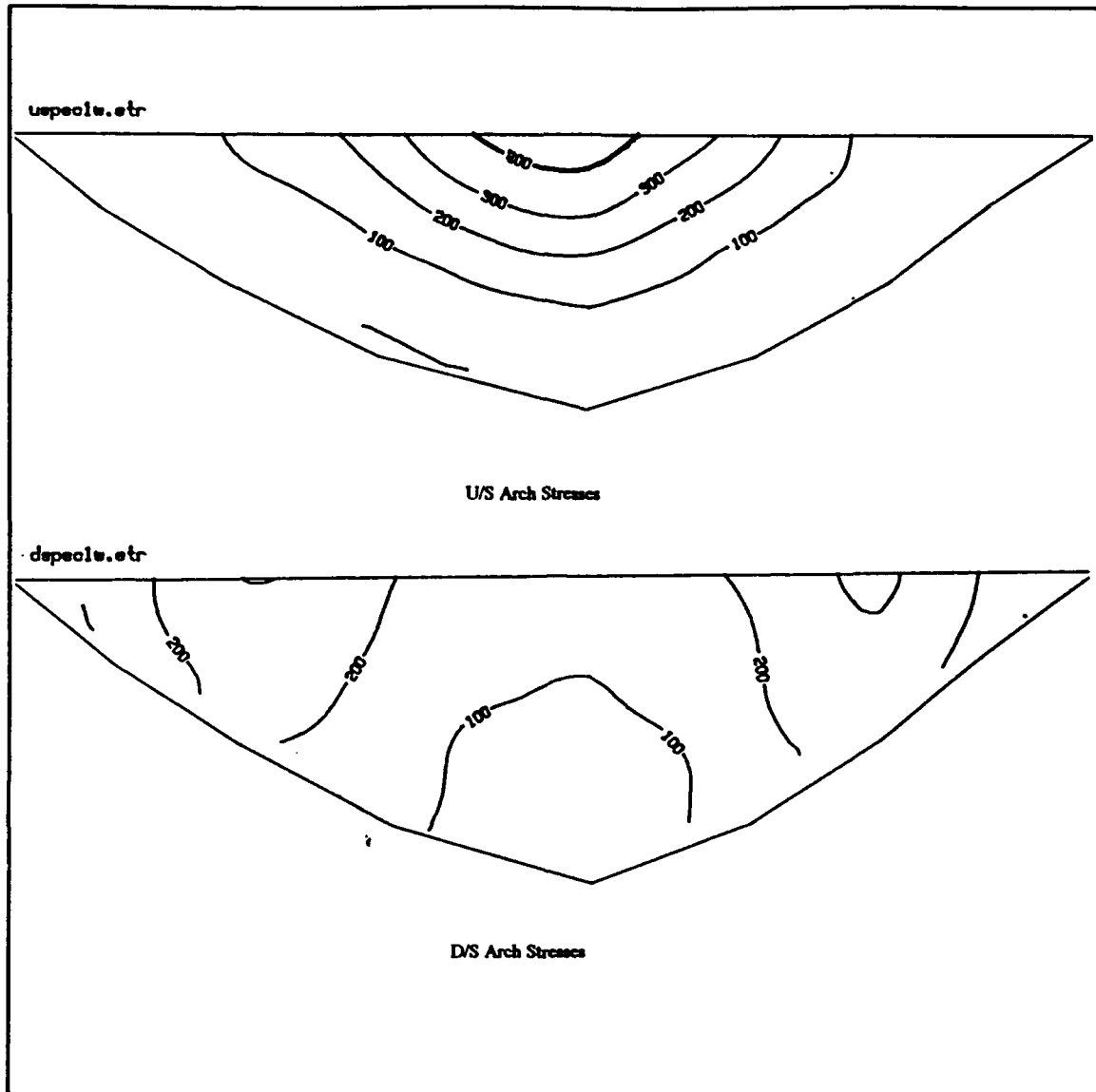
Figure A.22b  Dynamic Stresses Due to Response Spectrum
(Westergaard Added Mass)
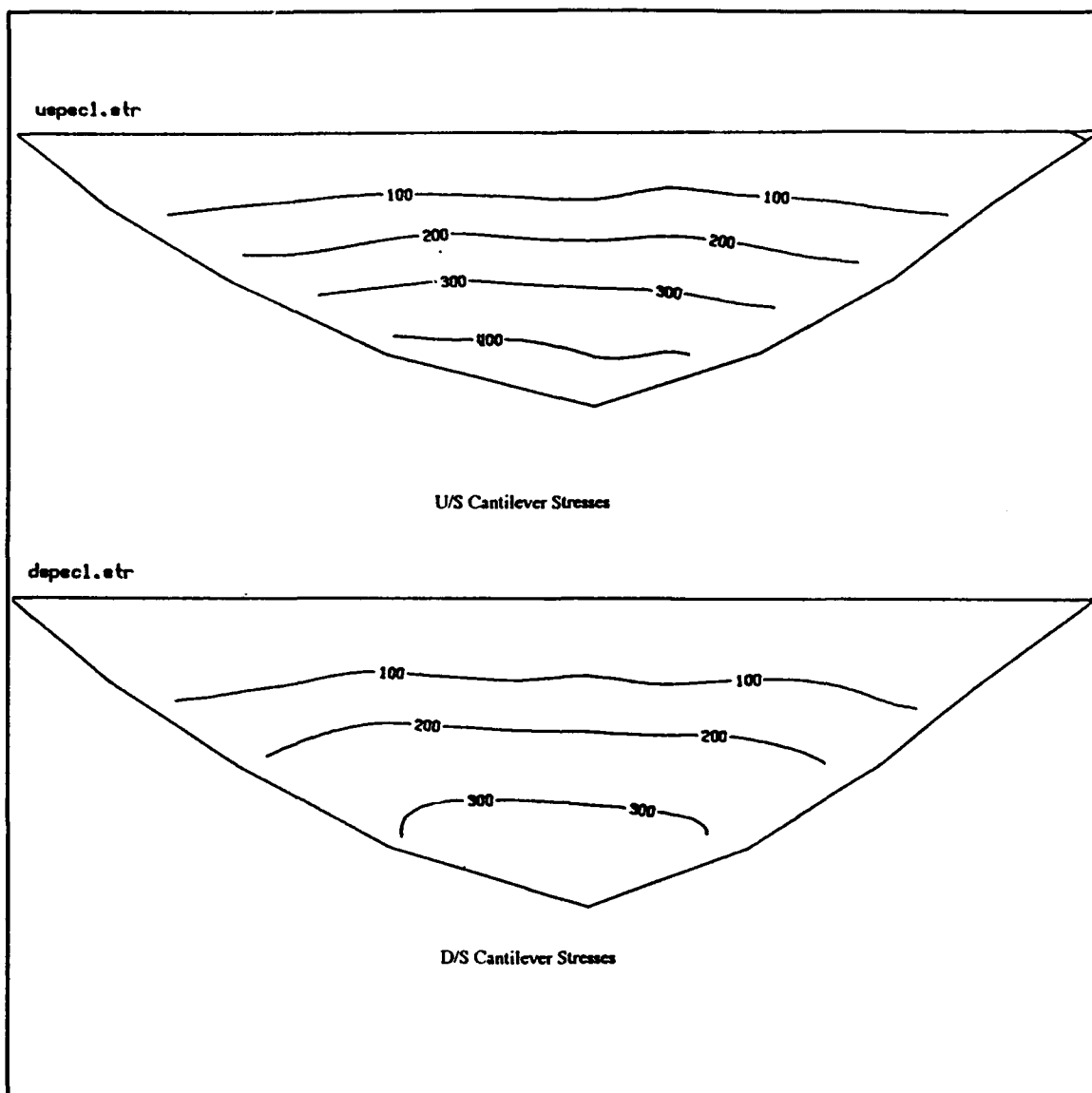
Figure A.23a  Dynamic Stresses Due to Response Spectrum (FE Added Mass)

uspeclw.str

U/S Cantilever Stresses

dspeclw.str

D/S Cantilever Stresses
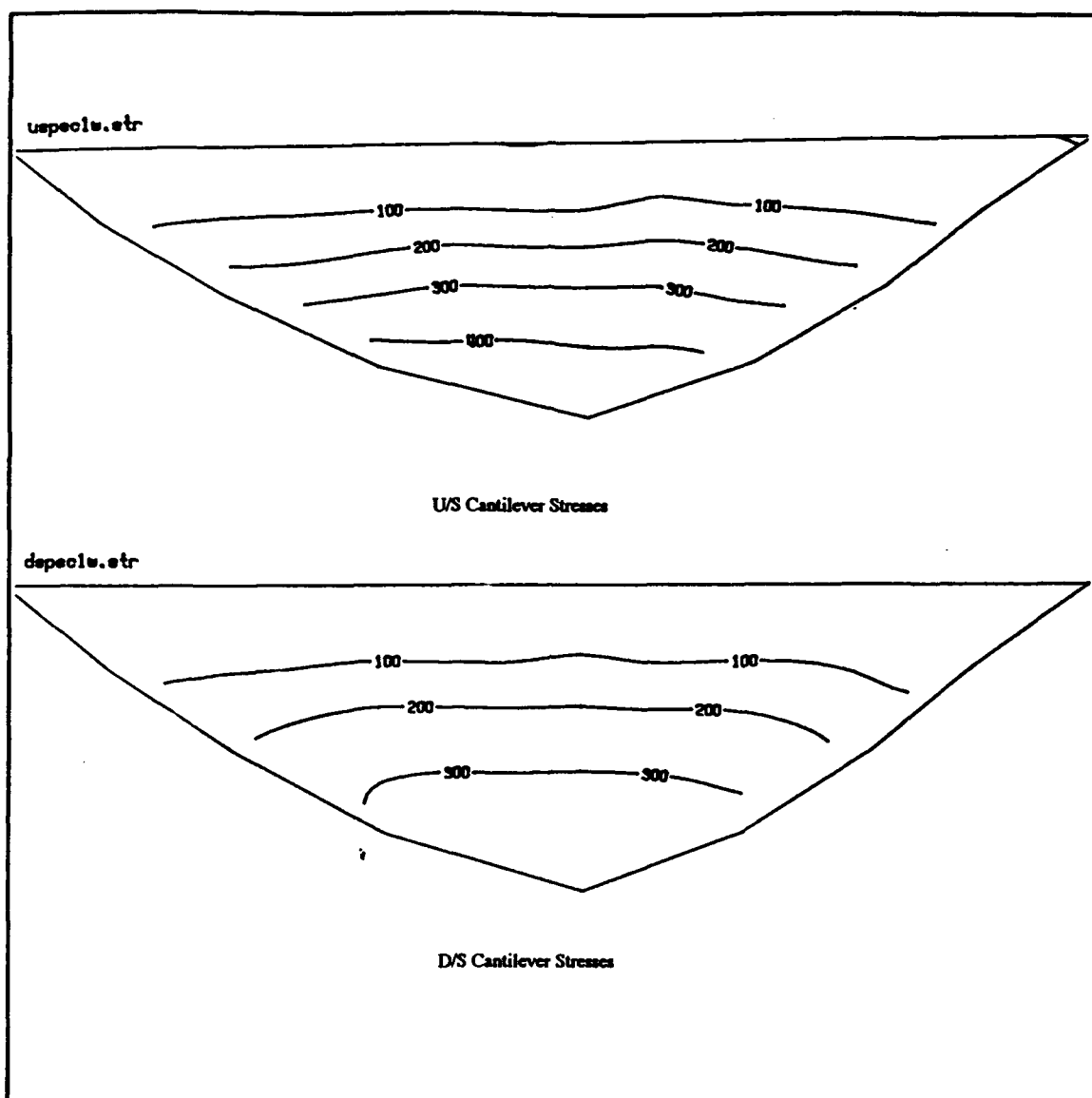
Figure A.23b  Dynamic Stresses Due to Response Spectrum
(Westergaard Added Mass)
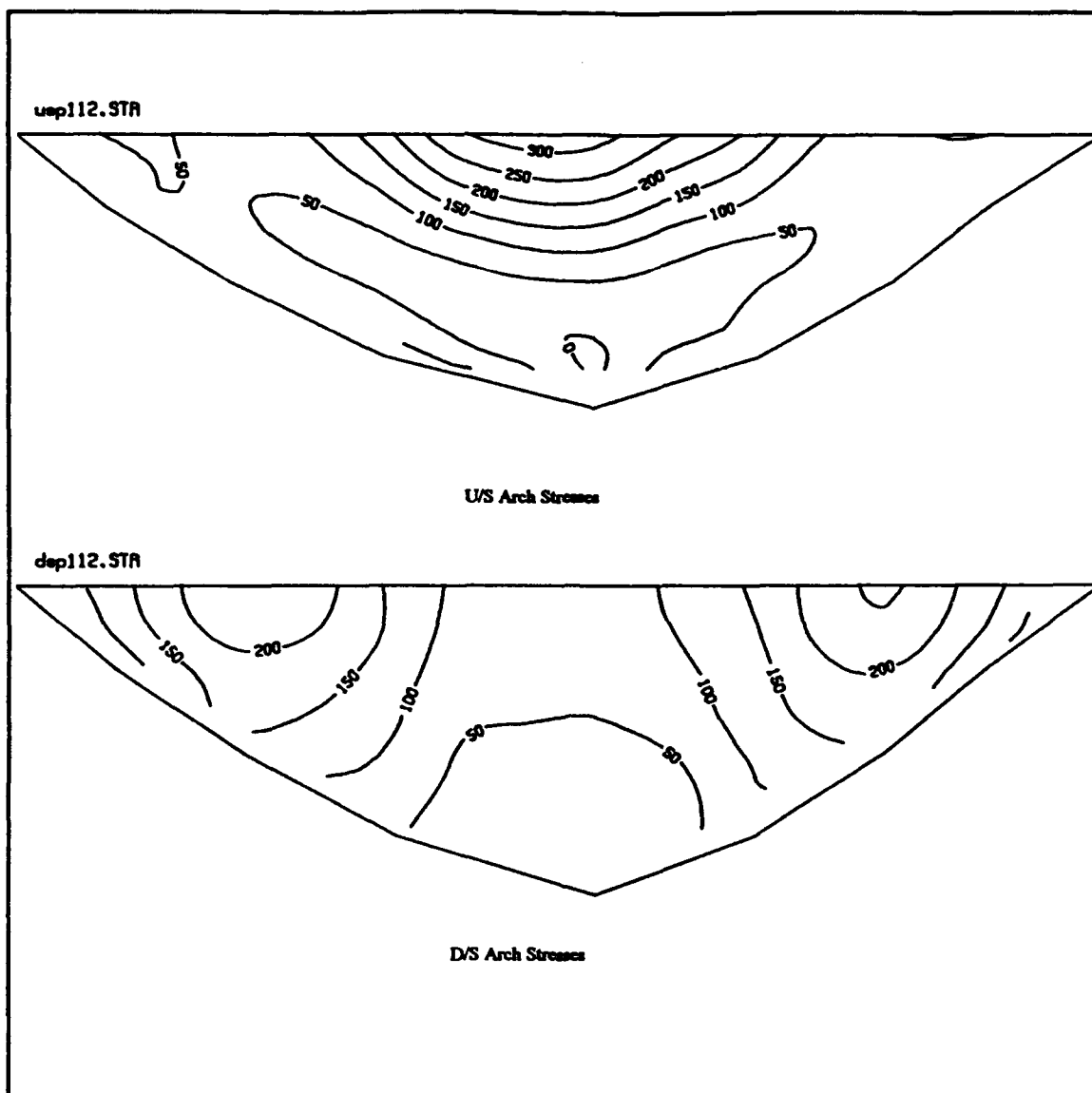
Figure A.24 Combined Stresses Due to Static + Response Spectrum

**uep112.str**

U/S Cantilever Stresses

**dep112.str**

D/S Cantilever Stresses
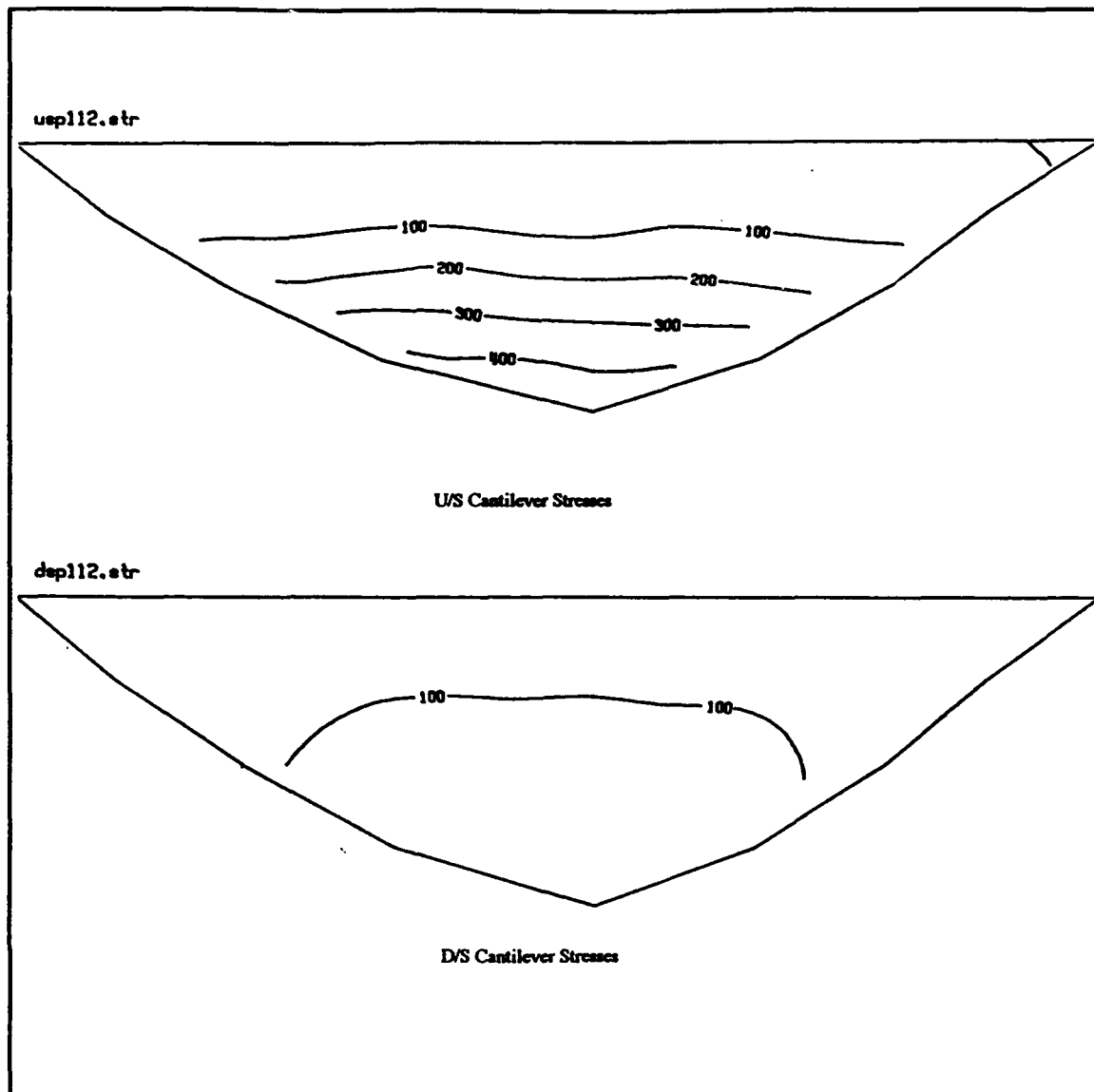
Figure A.25  Combined Stresses Due to Static + Response Spectrum

118
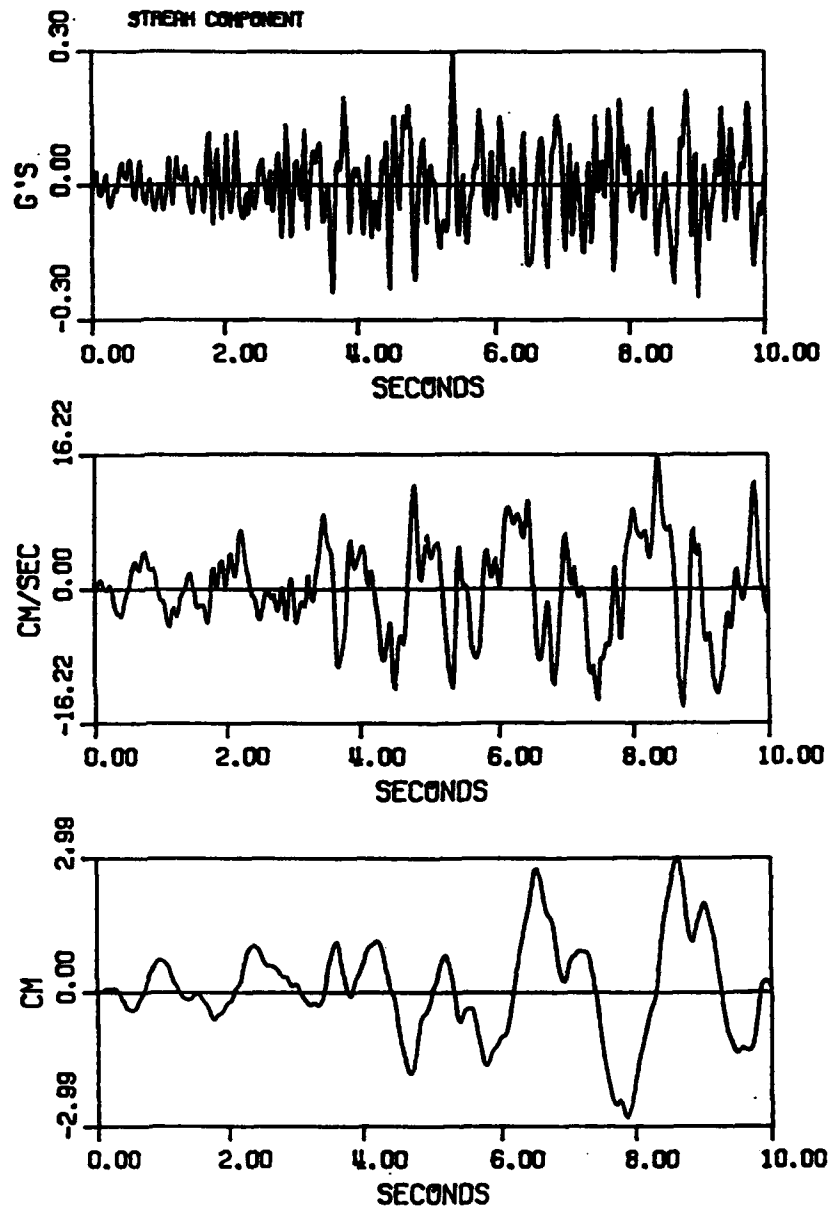
Figure A.26 Time-Histories of Longitudinal Component of Earthquake Motion

119
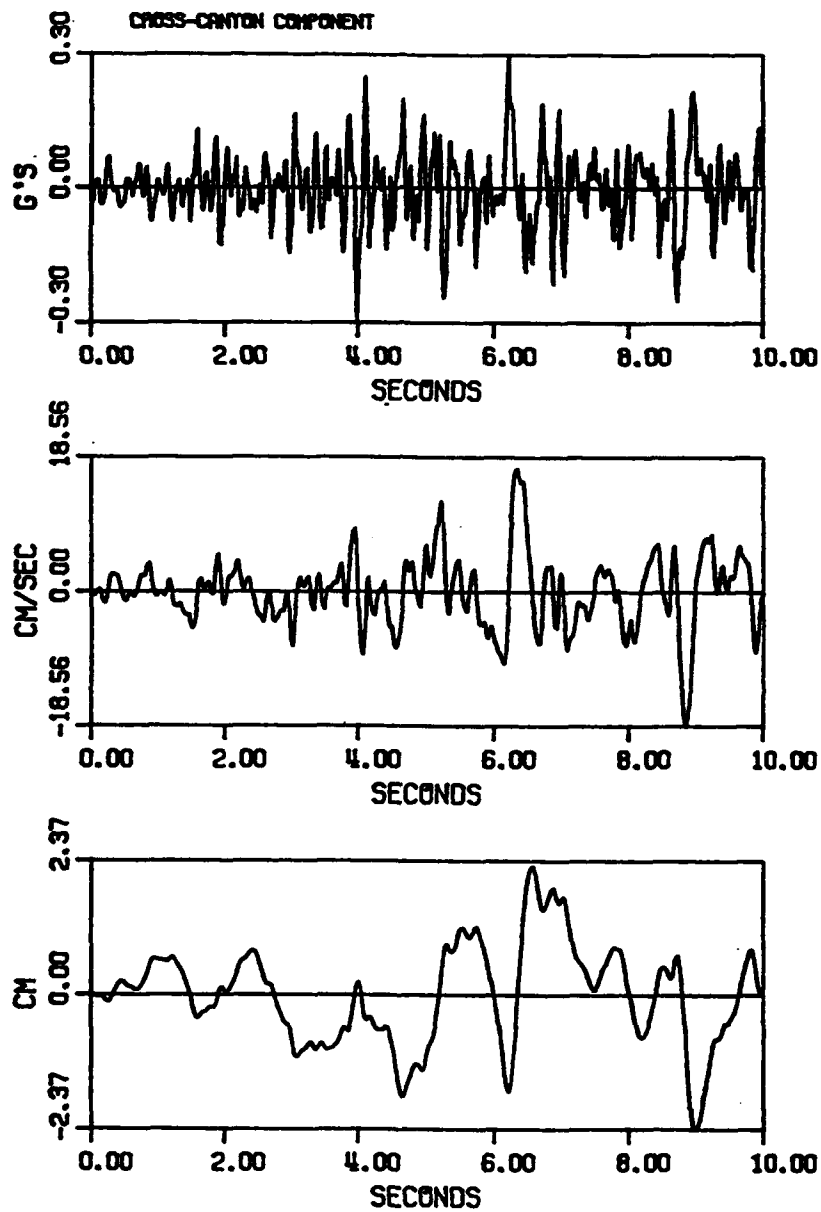
Figure A.27 Time-Histories of Transverse Component of Earthquake Motion
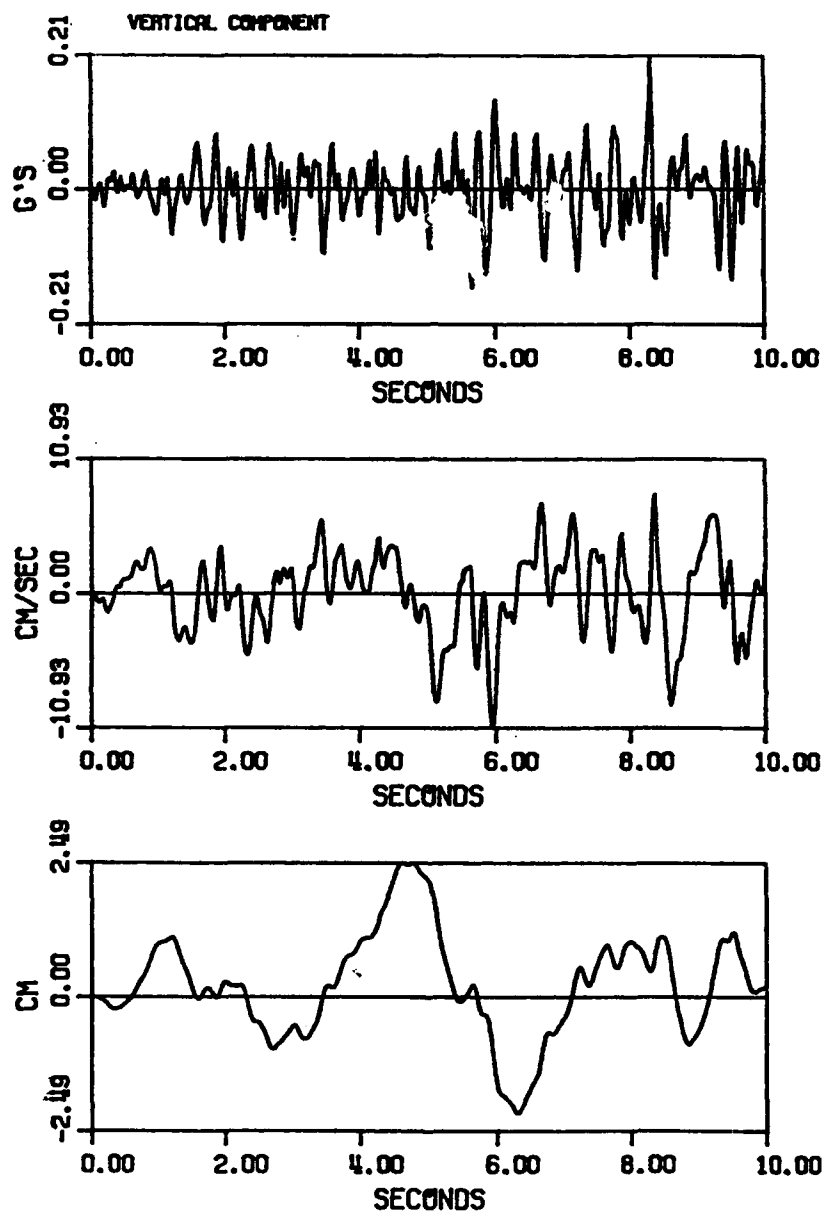
Figure A.28  Time-Histories of Vertical Component of Earthquake Motion

121
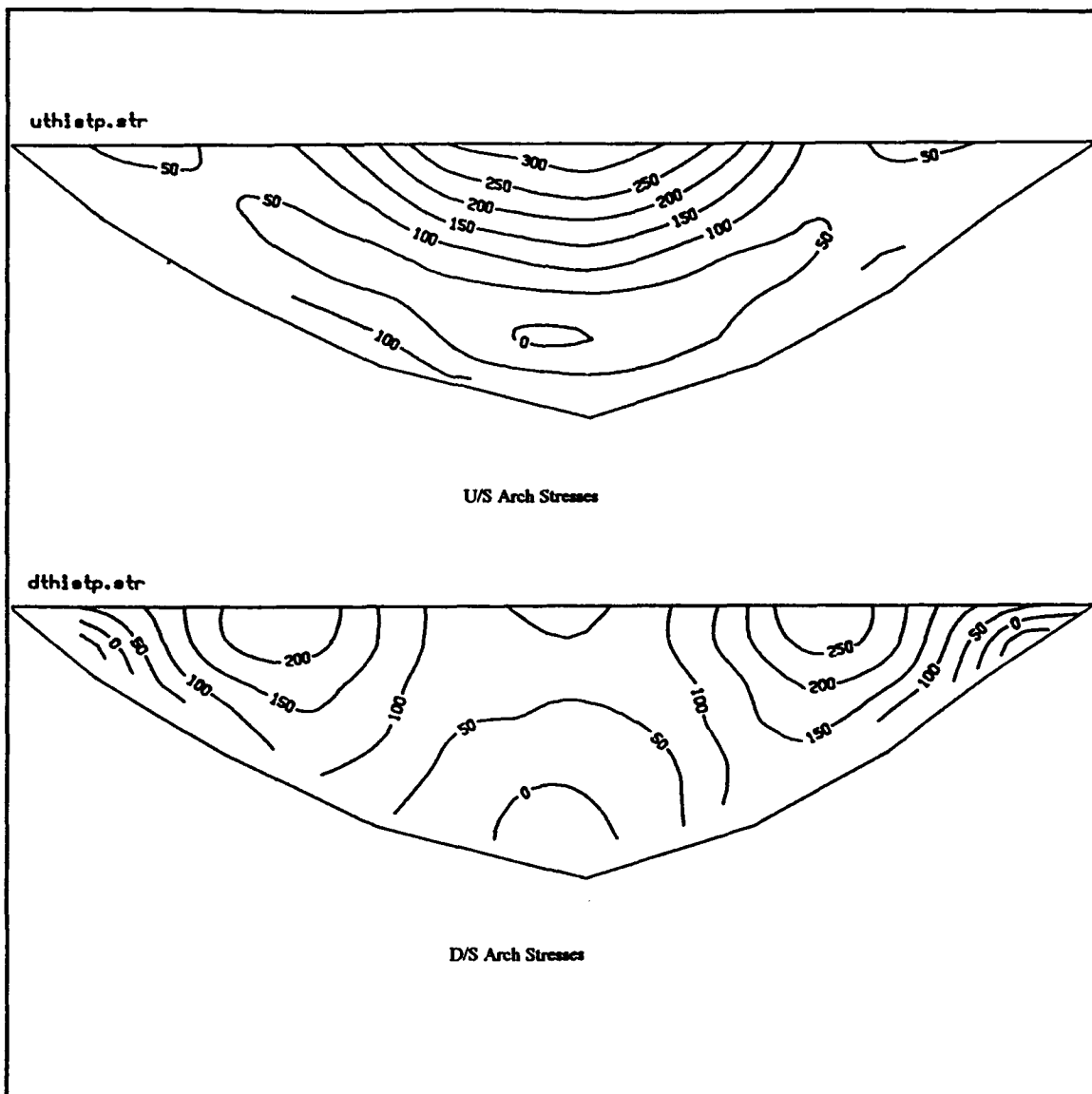
Figure A.29  Envelope of Tensile Arch Stresses
Static + Time-History
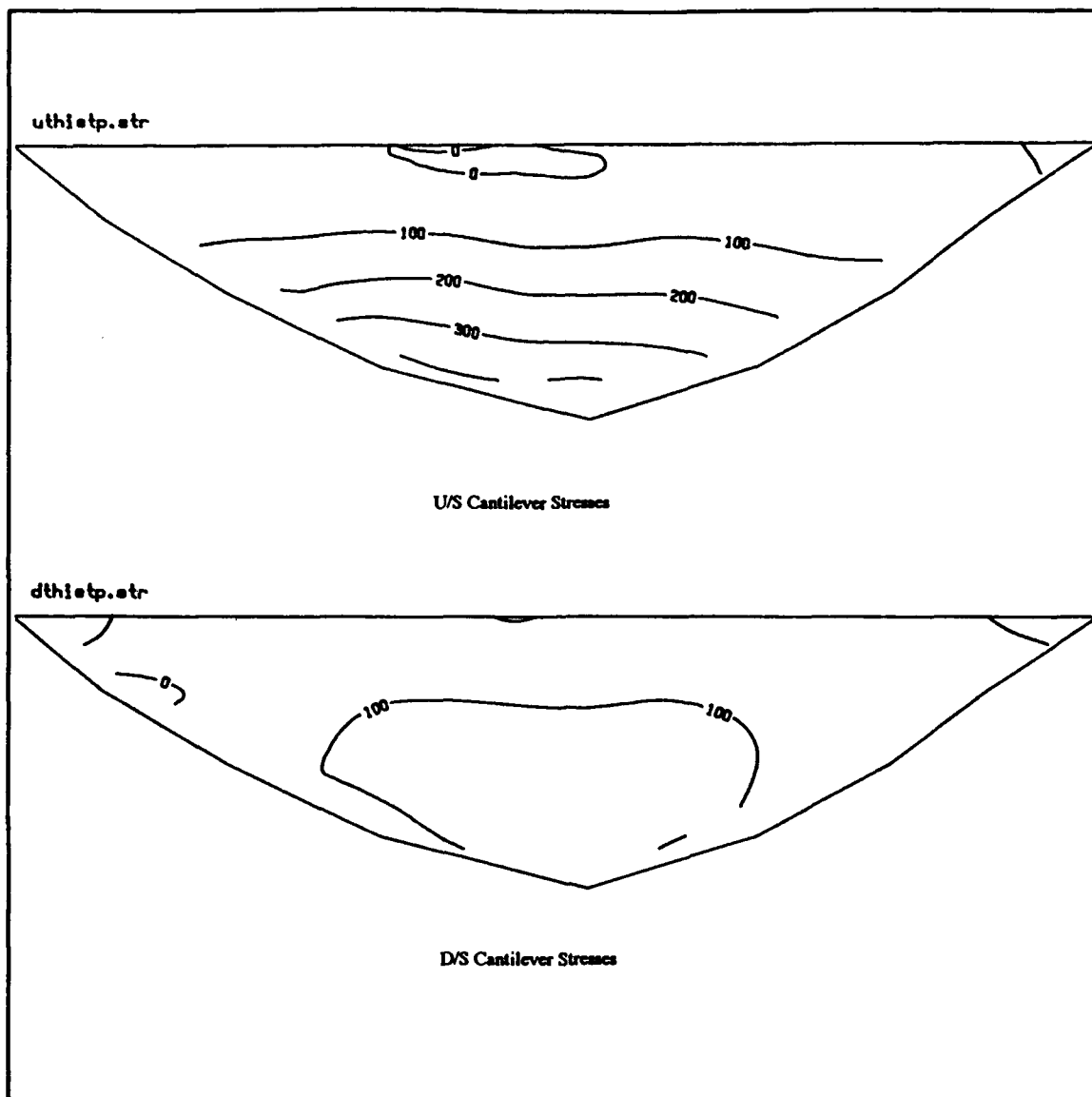
Figure A.30  Envelope of Tensile Cantilever Stresses
Static + Time-History
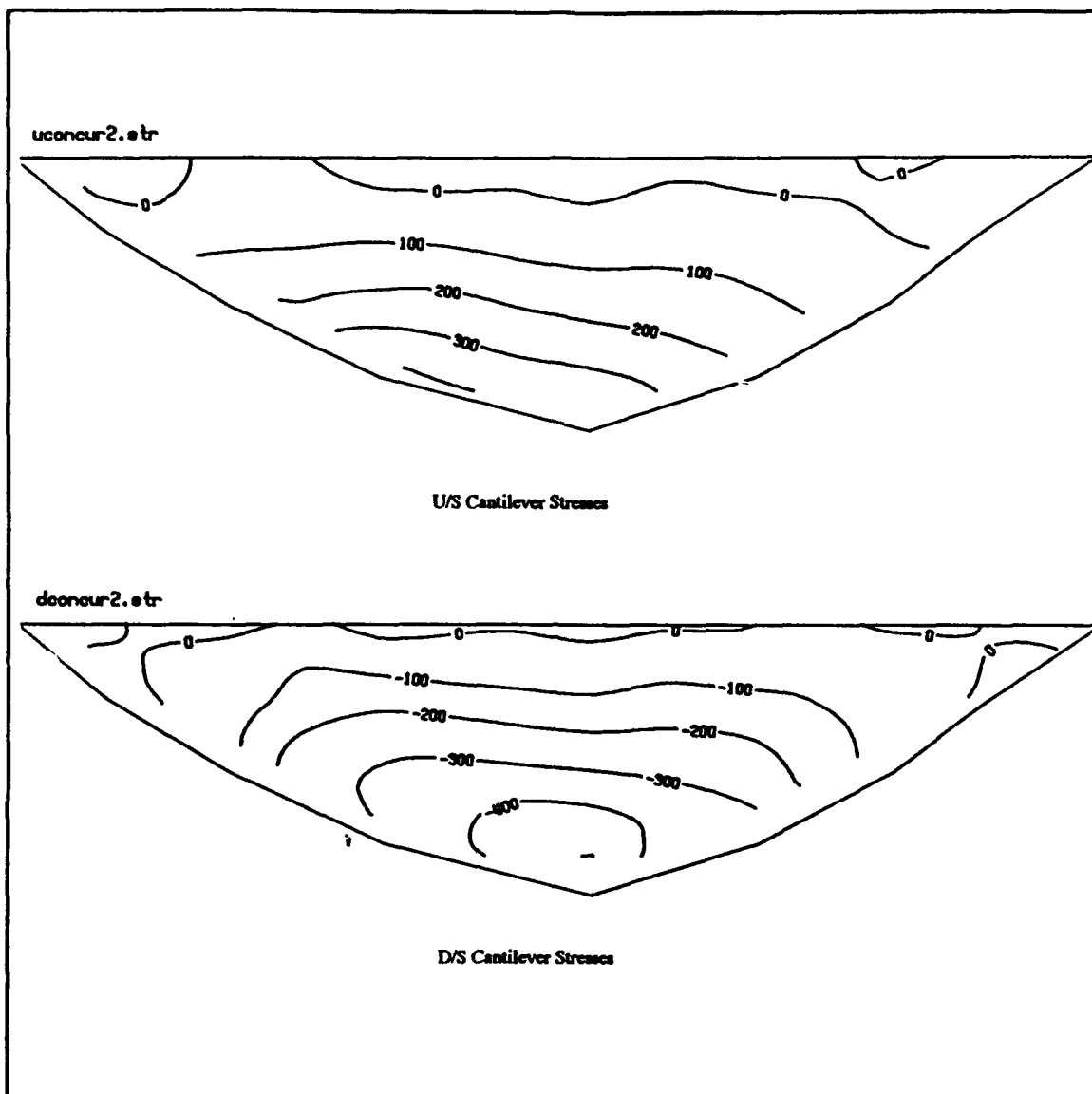
Figure A.31  Concurrent Cantilever Stresses at Time 7.06 sec.
Static + Time-History
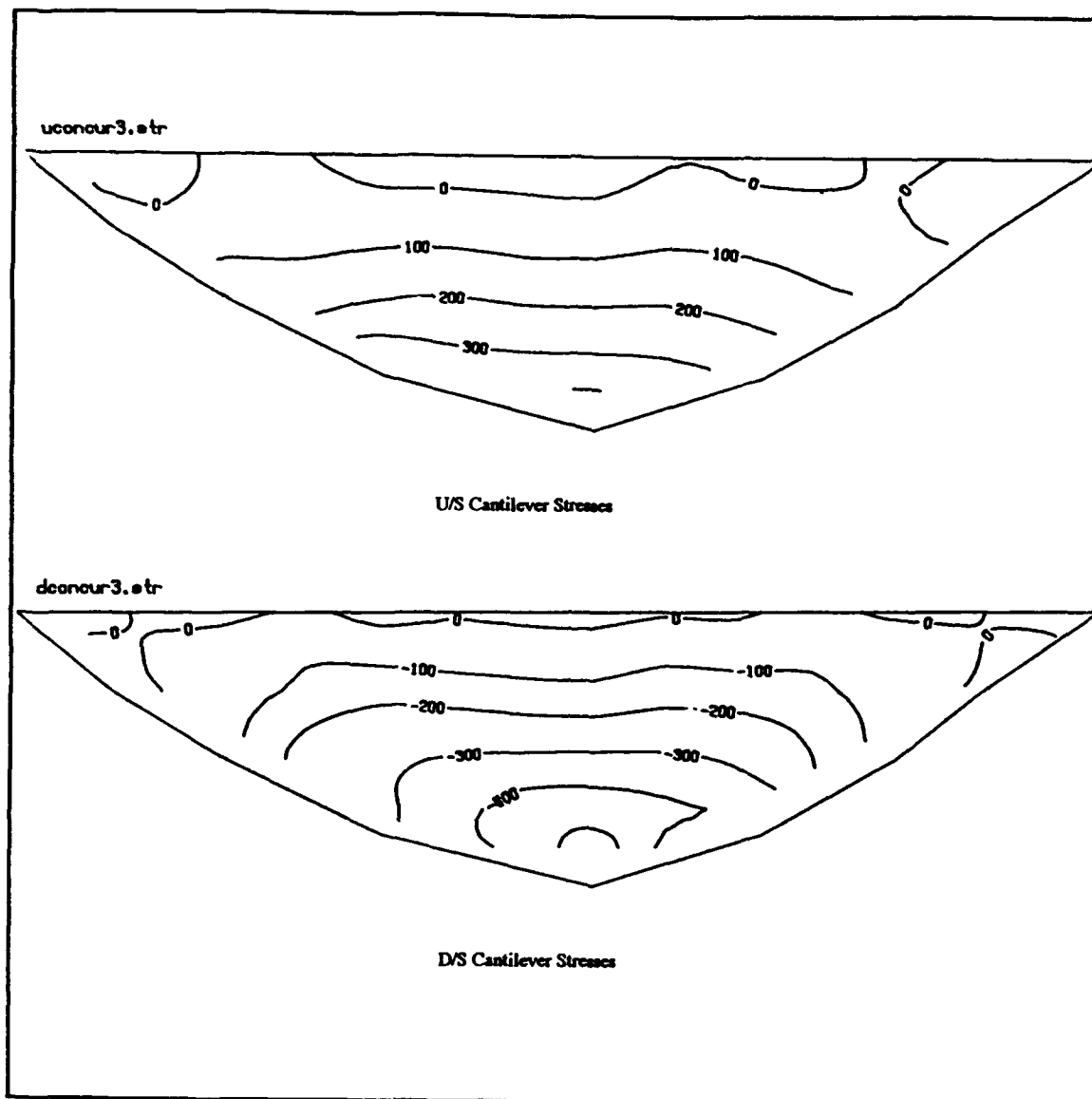
Figure A.32a  Concurrent Cantilever Stresses at Time 8.70 sec.
Static + Time-History
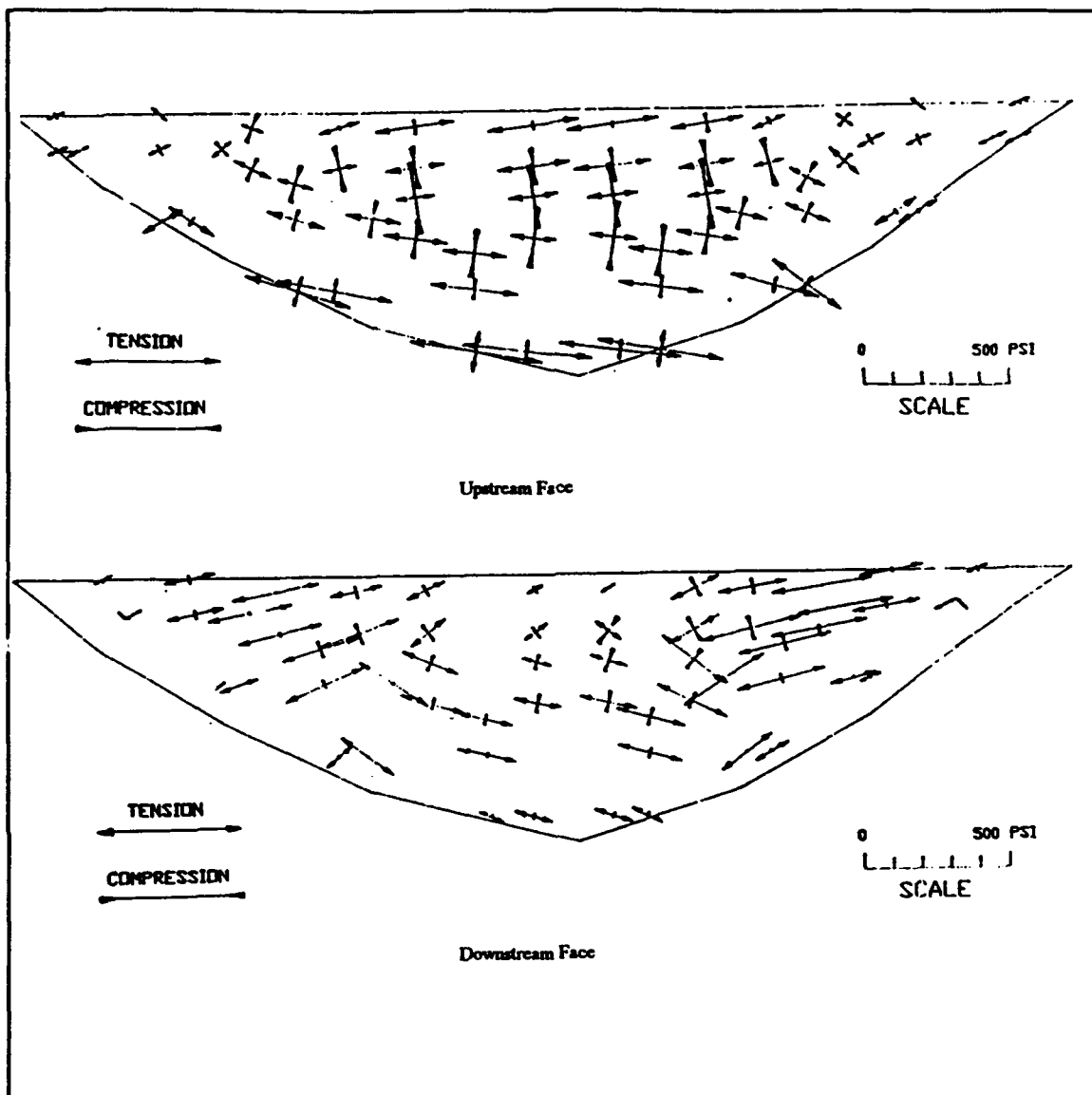
Figure A.32b  Envelope of Major Principal Stresses (Static + Time-History) with Corresponding Perpendicular Pairs
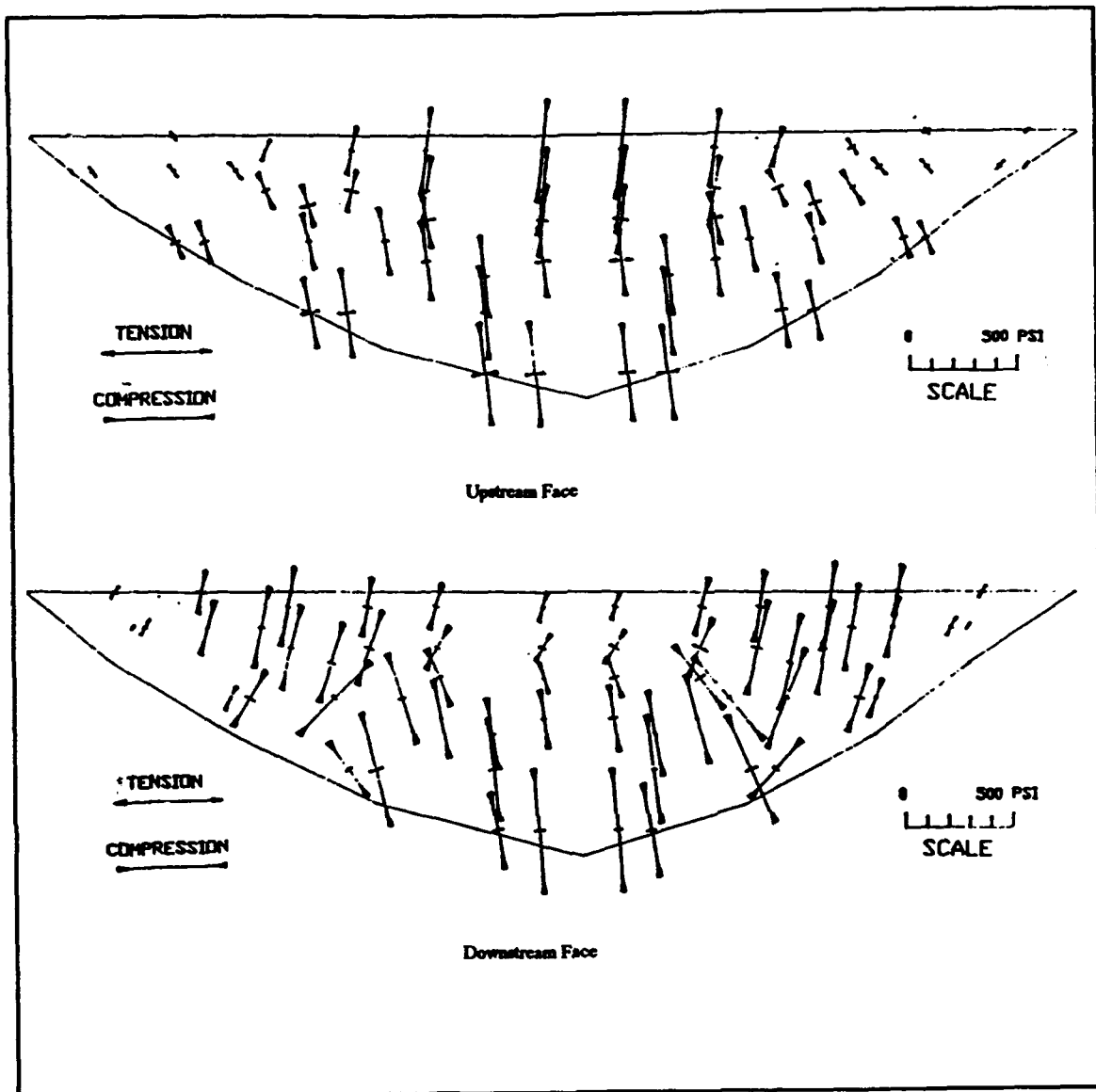
Figure A.32c Envelope of Minor Principal Stresses (Static + Time-History) with Corresponding Perpendicular Pairs
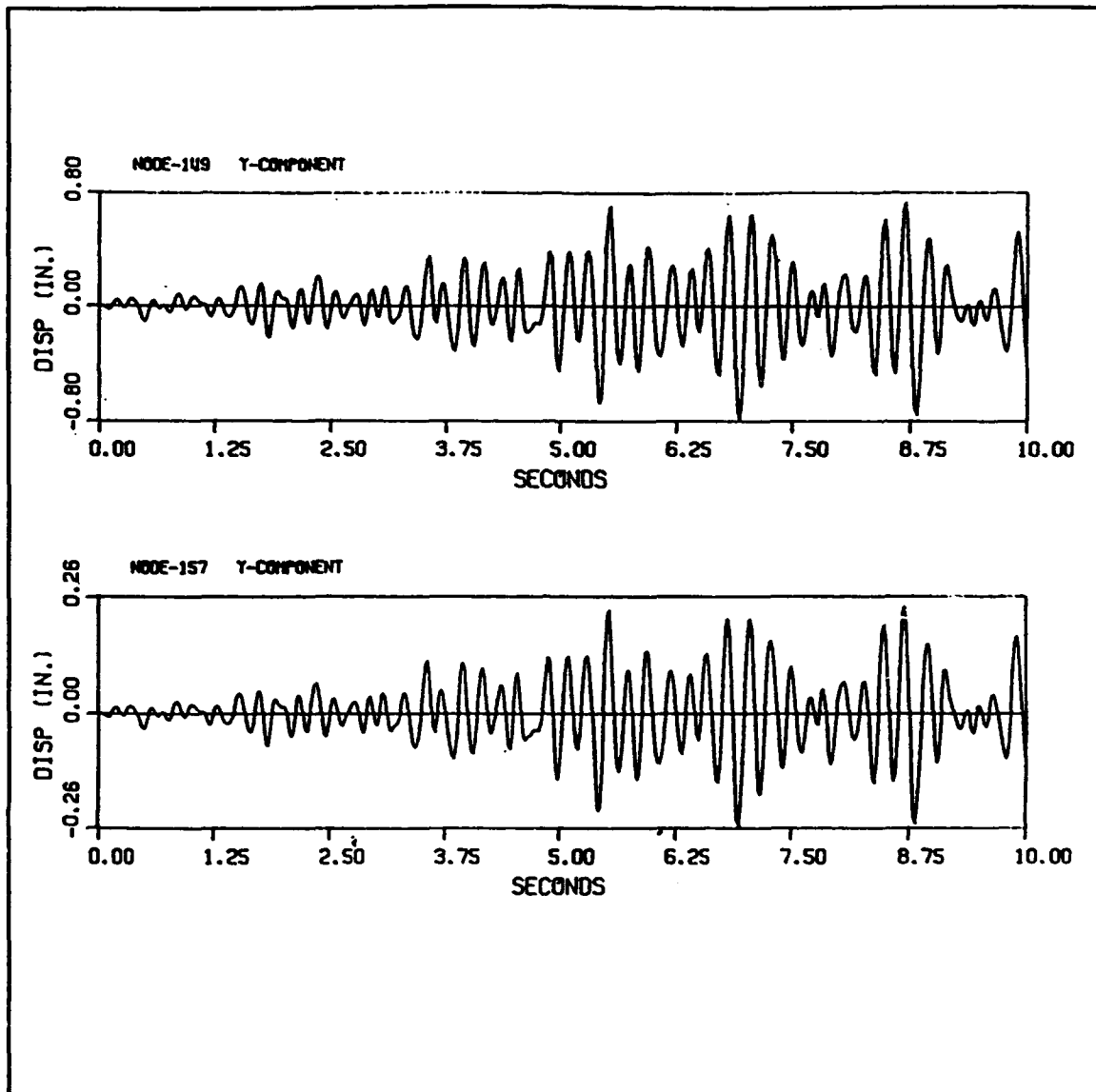
Figure A.33 Stream-Direction Displacement Time-Histories of Nodes 149 and 157, Located at Crest and Midheight Elevations, Respectively (Figure A.3)
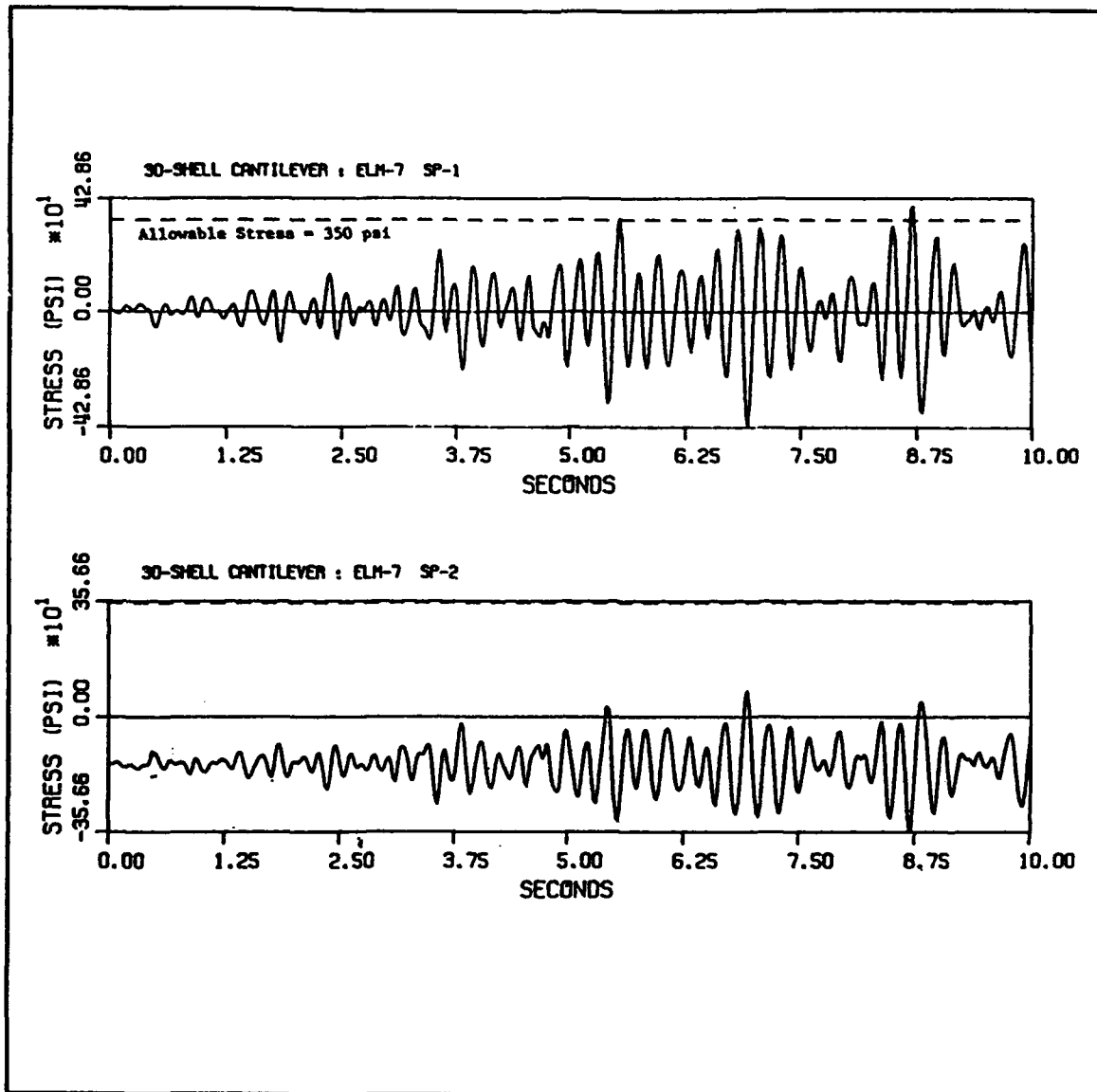
Figure A.34  Time-History of Cantilever Stresses for Stress Points 1 and 2 (Located on Upstream
Side and Downstream Side) of 3-D Shell Element No. 7 (Figures 6.9 and A.2)
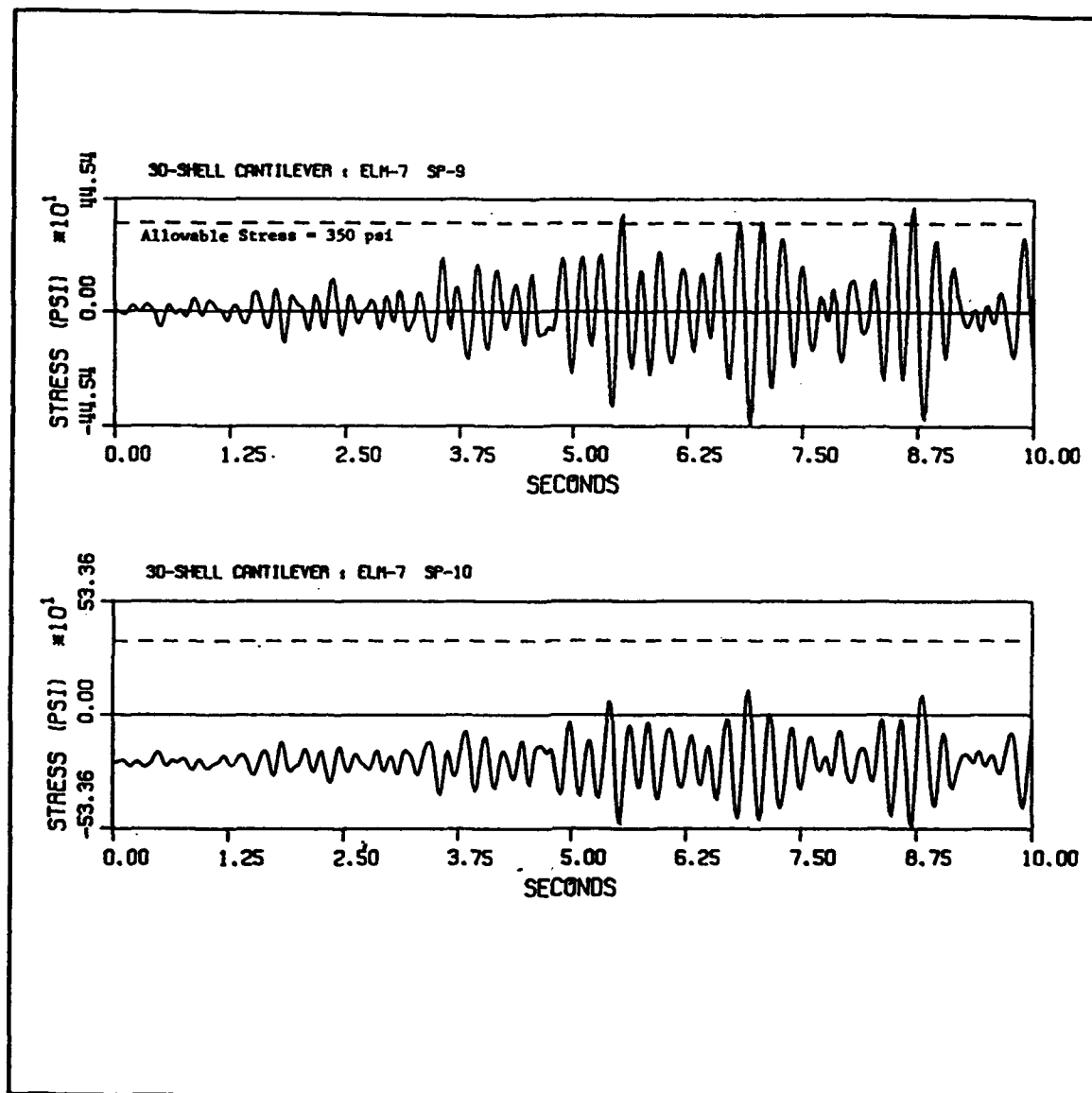
Figure A.35  Time-History of Cantilever Stresses for Stress Points 9 and 10 (Located on Upstream
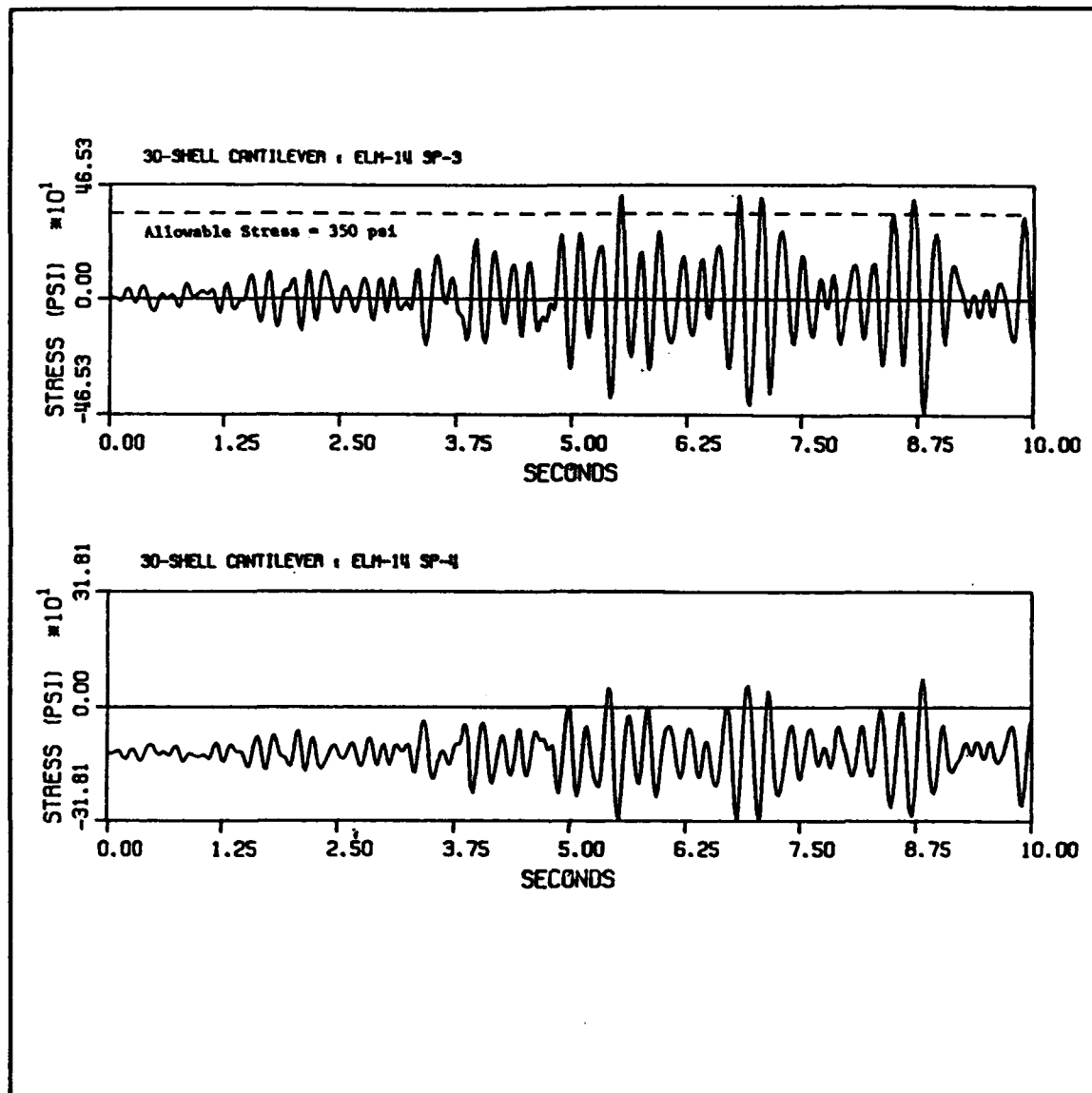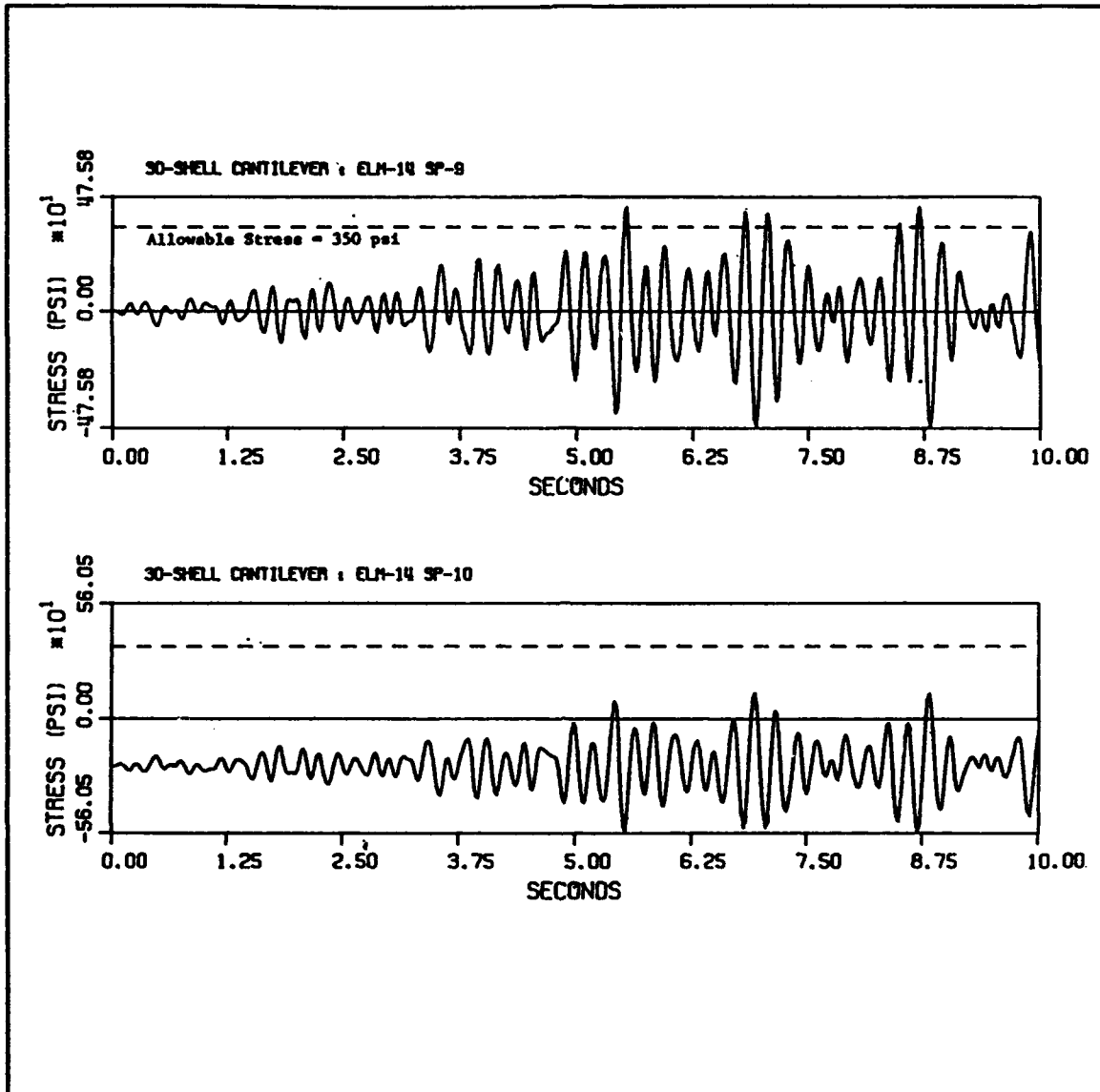Side and Downstream Side) of 3-D Shell Element No. 7 (Figures 6.2 and A.2)

130
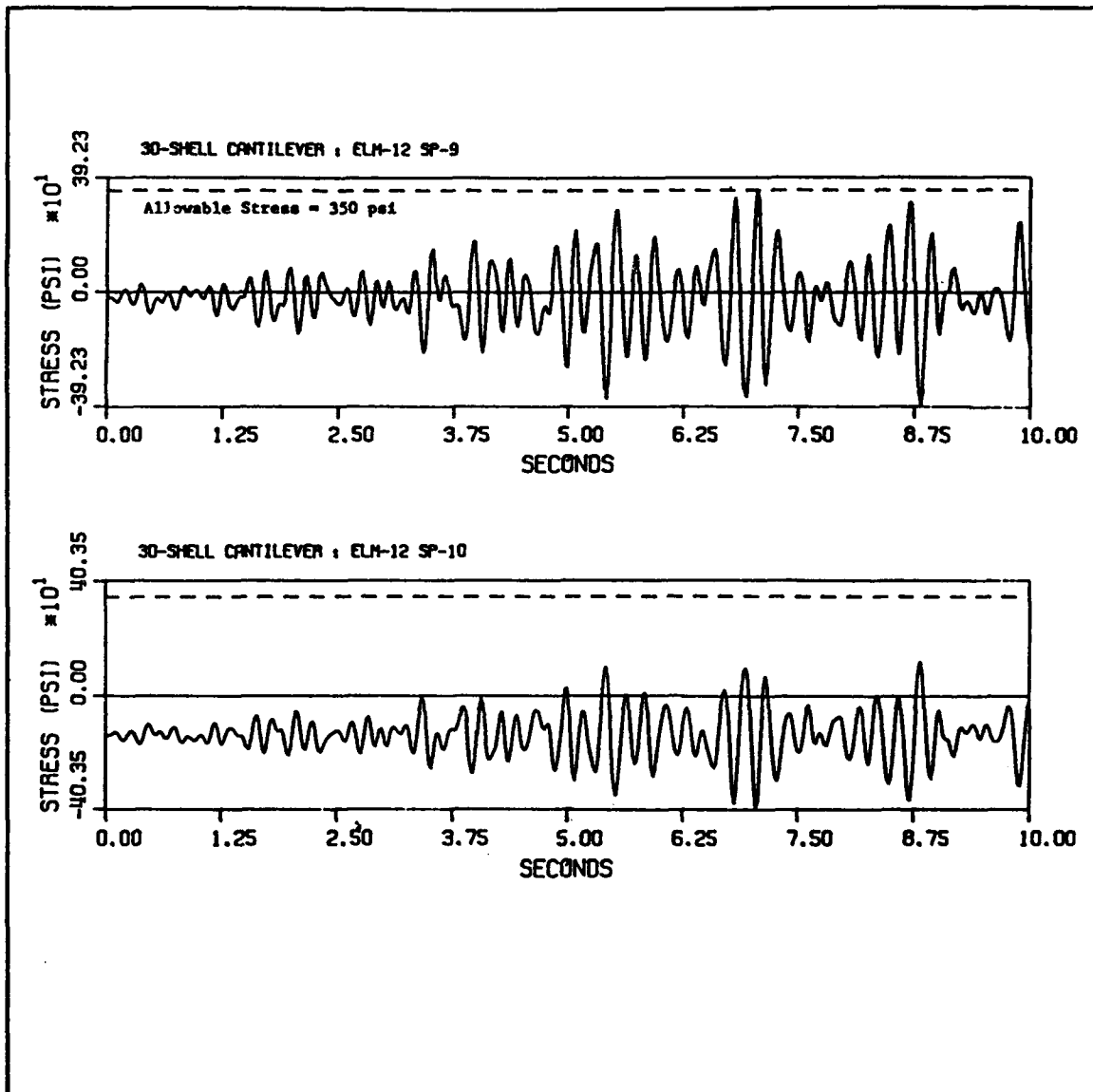
Figure A.36 Time-History of Cantilever Stresses for Stress Points 9 and 10 (Located on Upstream Side and Downstream Side) of 3-D Shell Element No. 12 (Figures 6.2 and A.2)

Figure A.37 Time-History of Cantilever Stresses for Stress Points 3 and 4 (Located on Upstream
Side and Downstream Side) of 3-D Shell Element No. 14 (Figures 6.2 and A.2)

Figure A.38  Time-History of Cantilever Stresses for Stress Points 9 and 10 (Located on Upstream
Side and Downstream Side) of 3-D Shell Element No. 14 (Figures 6.2 and A.2)

133

| 1. AGENCY USE ONLY (Leave blank) | 2. REPORT DATE August 1993 | 3. REPORT TYPE AND DATES COVERED Final report |
|---|---|---|

**4. TITLE AND SUBTITLE**

User's Manual — GDAP, Graphics-Based Dam Analysis Program Version 3.25

**5. FUNDING NUMBERS**

**6. AUTHOR(S)**

Yusof Ghanaat

**7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)**

Quest Structures
1900 Powell St., Suite 210
Emeryville, CA 94608

**8. PERFORMING ORGANIZATION REPORT NUMBER**

**9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)**

U.S. Army Corps of Engineers, Washington, DC 20314-1000;
U.S. Army Engineer Waterways Experiment Station, Information Technology
Laboratory, 3909 Halls Ferry Road, Vicksburg, MS 39180-6199

**10. SPONSORING/MONITORING AGENCY REPORT NUMBER**

Instruction Report ITL-93-3

**11. SUPPLEMENTARY NOTES**

See reverse.

**12a. DISTRIBUTION/AVAILABILITY STATEMENT**

Approved for public release; distribution is unlimited.

**12b. DISTRIBUTION CODE**

**13. ABSTRACT (Maximum 200 words)**

This manual describes the graphics-based dam analysis program (GDAP) which performs three-dimensional (3-D), finite element (FE) static and dynamic analyses of concrete arch and gravity dams on a desktop computer and provides graphic pre- and post-processing capabilities. The FE meshes of the concrete dam, foundation rock, and the impounded water are generated automatically from a limited amount of input data. Various two- (2-) and 3-D graphics are produced to examine the accuracy of the analytical models. The results of static and dynamic analyses are displayed in graphical forms for easy interpretation and evaluation. In particular, the GDAP post-processor automatically evaluates the response-history results and extracts the critical information for presentation and further evaluation.

**14. SUBJECT TERMS**

Arch dam
Dynamic analysis

**15. NUMBER OF PAGES**
141

**16. PRICE CODE**

| 17. SECURITY CLASSIFICATION OF REPORT UNCLASSIFIED | 18. SECURITY CLASSIFICATION OF THIS PAGE UNCLASSIFIED | 19. SECURITY CLASSIFICATION OF ABSTRACT | 20. LIMITATION OF ABSTRACT |
|---|---|---|---|

11. (Concluded).

The User's Manual is available from National Technical Information Service, 5285 Port Royal Road, Springfield, VA   22161.  The computer program is available to U.S. government employees only and can be obtained through the Engineering Computer Program Library at the U.S. Army Engineer Waterways Experiment Station, 3909 Halls Ferry Road, Vicksburg, MS   39180-6199.