

1. Report No. FHWA/TX-94/1182-4F		2. Government Accession No.		3. Recipient's Catalog No.	
4. Title and Subtitle GRAPHICALLY-ORIENTED ANALYSIS OF POST-TENSIONED SLAB BRIDGES ON MICROCOMPUTERS				5. Report Date February 1992, Revised: January 1994	
				6. Performing Organization Code	
7. Author(s) Paul N. Roschke and Kevin R. Pruski				8. Performing Organization Report No. Research Report 1182-4F	
9. Performing Organization Name and Address Texas Transportation Institute The Texas A&M University System College Station, TX 77843-3135				10. Work Unit No. (TRAIS)	
				11. Contract or Grant No. Study No. 0-1182	
12. Sponsoring Agency Name and Address Texas Department of Transportation Research and Technology Transfer Office P. O. Box 5080 Austin, TX 78763-5080				13. Type of Report and Period Covered Final Report: September 1987 - August 1991	
				14. Sponsoring Agency Code	
15. Supplementary Notes Research performed in cooperation with the Texas Department of Transportation and the U.S. Department of Transportation, Federal Highway Administration. Research Study Title: Evaluation of Factors Affecting Slabs Due to Localized Post-Tension Forces					
16. Abstract <p>This is the fourth in a series of reports documenting a research program aimed at detailed investigation of a new type of bridge structure that has a moderately thick, post-tensioned, concrete slab resting directly on columns without bent caps. In this program, two scaled laboratory model, named Model One and Model Two, are tested in the laboratory along with instrumentation of a full-scale 3-span bridge. Analysis of this class of bridge structure is carried out by means of a nonlinear finite element analysis code that has been specially designed to run on microcomputers. This report relates to the analysis code and is intended to serve as a user manual for design engineers.</p> <p>A suite of computer codes, mnemonically titled <i>TEXSLAB</i>, automates input of bridge geometry and loads, performs finite element simulation to determine deflections, strains, and stresses in the slab and tendons, and provides graphical output of salient quantities for the designer. A user-oriented system of menu-driven interfaces makes generation of the input model relatively simple. In addition, graphical representation of many input parameters, such as tendon profiles, allows error-checking before execution of the analysis module.</p> <p>Full program operation takes place on a microcomputer. This package is developed using FORTRAN 77 as the primary coding language, C as a secondary language for operating system function calls, and a commercially available graphics package for rapid visualization of results. <i>TEXSLAB</i> runs under OS/2 version 1.1, 1.2, 1.3, 2.0, or 2.1 with the DOS compatibility box active. File development and program execution take place in OS/2, while graphic model confirmation and examination of analysis results occur in DOS.</p>					
17. Key Words Banded Tendon, Bridge, Computer, Concrete, Deflection, Failure, Finite Element, Graphics, Nonlinear, Plate, Post-tensioning, Prestressing, Shear, Slab, Strain, Stress, Transverse Stressing			18. Distribution Statement No restrictions. This document is available to the public through NTIS: National Technical Information Service 5285 Port Royal Road Springfield, VA 22161		
19. Security Classif.(of this report) Unclassified		20. Security Classif.(of this page) Unclassified		21. No. of Pages 117	22. Price

**GRAPHICALLY-ORIENTED ANALYSIS OF
POST-TENSIONED SLAB BRIDGES ON MICROCOMPUTERS**

by

Paul N. Roschke
Assistant Research Engineer

and

Kevin R. Pruski
Graduate Research Assistant

Research Report 1182-4F
Research Study Number 0-1182
Research Study Title: Evaluation of Factors Affecting Slabs
Due to Localized Post-Tension Forces

Sponsored by the
Texas Department of Transportation
In Cooperation with
U.S. Department of Transportation
Federal Highway Administration

February 1992
Revised: January 1994

TEXAS TRANSPORTATION INSTITUTE
The Texas A&M University System
College Station, Texas 77843-3135

IMPLEMENTATION

Findings of this study are available for immediate implementation. The computer program *TEXSLAB* can be used to aid in design of flat slab concrete bridges that are bidirectionally post-tensioned. Material and geometrical nonlinearities can be studied for concentrated and distributed loads on the slab. The code has been validated by means of two laboratory models and an extensive study of a field bridge. Graphical results are available to the designer before and after a finite element analysis is carried out. The code can be run on an IBM compatible microcomputer which is readily available to designers of post-tensioned bridges.

Potential benefits include increased accuracy in the analysis of slab bridges. A higher level of confidence in the analysis can lead to savings in materials and a more consistent factor of safety during design.

In order to translate the research product into applicable form for use by TxDOT personnel, the code needs to be loaded on a microcomputer and the designer needs to study this report in order to become familiar with the fundamental assumptions of the code and the input and output limitations and capabilities.

This report concentrates on one phase of a large study and needs to be read in the context of the other companion reports. Emphasis here is on the analysis software. Complementary work on two laboratory models and a full-scale instrumented bridge (see reports 1182-1, 1182-2, and 1182-3) will be helpful for designers who analyze these structures.

No new specifications, standards, materials, or equipment need to be developed as a result of the study.

DISCLAIMER

The contents of this report reflect the views of the authors who are responsible for the opinions, findings, and conclusions presented herein. The contents do not necessarily reflect the official views or policies of the Texas Department of Transportation or the Federal Highway Administration. This report does not constitute a standard, specification, or regulation; it is not intended for construction, bidding, or permit purposes.

The engineer in charge of this project is Dr. Paul N. Roschke, who is a registered professional engineer in the State of Texas (Serial Number 53889).

ACKNOWLEDGMENTS

This study was conducted under a cooperative program between the Texas Transportation Institute, the Texas Department of Transportation, and the Federal Highway Administration. Randy Cox and Tim Bradberry worked closely with the researchers, and their comments and suggestions are appreciated.

TABLE OF CONTENTS

	<i>Page</i>
List of Figures	xi
List of Tables	xii
Summary	xiii
1. Background and Significance of Work.....	1
2. Introduction	3
3. Setting Up <i>TEXSLAB</i>.....	5
3.1 Hardware and Software Requirements.....	5
3.1.1 Operating System	5
3.1.2 Extended Memory.....	5
3.1.3 Hard Disk Space.....	5
3.2 Installation.....	6
3.3 Configuration.....	8
3.4 Execution.....	8
4. Description of Program	9
4.1 File Menu	9
4.1.1 New	10
4.1.2 Open	10
4.1.3 Change Name.....	10
4.1.4 Directory	10
4.1.5 Exit.....	11
4.2 Edit Menu	11
4.3 Transform Menu	12
4.4 View Menu.....	12
4.4.1 Preview.....	12
4.4.2 Postview	13
4.5 Analyze Menu.....	14
4.5.1 Run	14
4.5.2 Kill.....	15
5. Input Editor.....	17
5.1 Geometric Parameters	17
5.2 Concrete	21
5.3 Reinforcing Steel.....	24

5.4 Prestressing Steel	26
5.5 Prestressing Tendons	28
5.6 Loads	32
5.7 Convergence Parameters	38
6. Output Editor	41
6.1 General Control	41
6.2 Nodal Control	43
6.3 Element Control	44
6.4 Reinforcement/Prestress Control	47
7. Sample Session	49
7.1 File	49
7.2 Edit	49
7.3 Transform	50
7.4 View	50
7.4.1 Preview	51
7.4.2 Postview	54
7.5 Analyze	56
8. Work Files	59
8.1 Default Microfiles	59
8.2 Input Microfiles	59
8.3 Transform Output Files	60
8.4 Analysis Output Files	61
Appendix I. Operating <i>TEXSLAB</i> in OS/2	63
OS/2 Version 1.1	63
OS/2 Versions After Version 1.1	64
Appendix II. Background and Commands for SuperView	67
Appendix III. Example Microfiles	81
Appendix IV. Example FEM Input Data File	91
Appendix V. Example FEM Output Data File	95
References	103

LIST OF FIGURES

<i>Figure</i>		<i>Page</i>
1	Example of Post-Tensioned Continuous Flat Slab Bridge	1
2	Geometric Representation of Input Model	18
3	Haunched Bridge Slab.....	20
4	Concrete Layer System.....	23
5	Reinforcing Steel Layer System	25
6	Stress versus Strain for a Prestressing Tendon.....	28
7	Tendon Profile	30
8	Specification of Prestressing Tendon Groups.....	30
9	Example of Lane Loading.....	35
10	Truck Loading.....	37
11	Example of Line Loads.....	39
12	Sign Convention for Components of Stress and Strain.....	46
13	Plan View of Sample Problem	52
14	Profile View of Sample Problem	54
15	Stress Dither of Sample Problem.....	56
16	Displacement Dither of Sample Problem	57

LIST OF TABLES

<i>Table</i>		<i>Page</i>
1	Examples of Disk Space and Execution Times	14
2	Iteration Codes for Reformulation of Stiffness Matrices	40
3	Description of Output Control Codes for Nodes	43
4	Description of Output Control Codes for Elements.....	47

SUMMARY

This is the fourth in a series of reports documenting a research program aimed at detailed investigation of a new type of bridge structure that has moderately thick, post-tensioned, concrete slabs resting directly on columns without bent caps. In this program, two scaled laboratory models, named Model One and Model Two, are tested in the laboratory along with instrumentation of a full-scale 3-span bridge. Analysis of this class of bridge structure is carried out by means of a nonlinear finite element analysis code that has been specially designed to run on microcomputers. This report relates to the analysis code and is intended to serve as a user manual for design engineers.

A suite of computer codes, mnemonically titled *TEXSLAB*, automates input of bridge geometry and loads, performs finite element simulation to determine deflections, strains, and stresses in the slab and tendons, and provides graphical output of salient quantities for the designer. A user-oriented system of menu-driven interfaces makes generation of the input model relatively simple. In addition, graphical representation of many input parameters, such as tendon profiles, allows error-checking before execution of the analysis module.

Full program operation takes place on a microcomputer. This package is developed using FORTRAN 77 as the primary coding language, C as a secondary language for operating system function calls, and a commercially available graphics package for rapid visualization of results. *TEXSLAB* runs under OS/2 version 1.1, 1.2, 1.3, 2.0, or 2.1 with the DOS compatibility box active. File development and program execution take place in OS/2, while graphic model confirmation and examination of analysis results occur in DOS.

1. BACKGROUND AND SIGNIFICANCE OF WORK

In recent years, a new class of bridge structure has been developed and constructed. Instead of using the typical prestressed girder and reinforced concrete slab, Texas Department of Transportation (TxDOT) is successfully building continuous, moderately long-span post-tensioned slab bridges. These structures are specified when a special need for minimum vertical clearance arises. The flat slab bridge offers some distinct advantages over alternative designs. Primarily, the depth of this structure is less than that of others. Depth of structure becomes important at locations where clearance becomes a problem and methods of increasing the clearance are not cost-effective. For example, these situations exist where new highways are built through existing roadways or interchanges and space is at a premium. In such cases, the height of the embankments is limited and, thus, the depth of the structure is bounded. Post-tensioned flat slab bridges also offer an aesthetically pleasing design alternative. They are relatively thin (approximately 30 in. [76.2 cm]) structures that are popular because of low maintenance and good durability. An example of such a bridge is shown in Fig. 1.

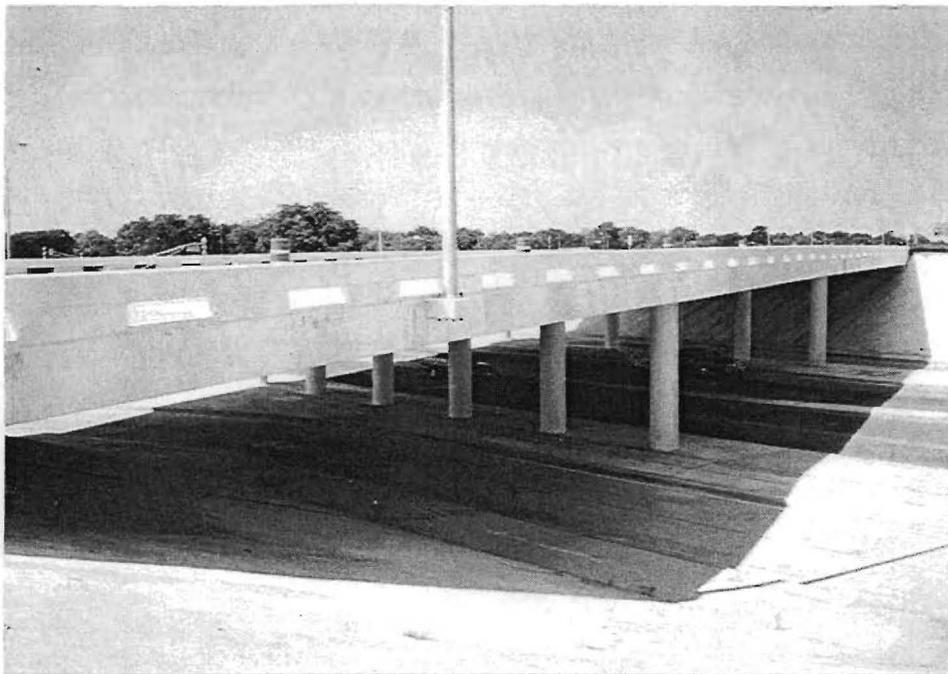


FIG. 1. Example of Post-Tensioned Continuous Flat Slab Bridge

Complementing the new state-of-the-art structure are sophisticated methods of analysis that are available as well. However, present methods of designing post-tensioned flat slab bridges generally follow techniques developed for the more ubiquitous beam bridges. Simplifications are made by idealizing this structure into a strip (beam). Effects of skew, if present, are neglected, and additional benefits of two-way plate action that is associated with slabs are not considered. In addition, the complexities associated with prestressing, the nonlinear and time-dependent nature of concrete, and the possibility that biaxial states of stress exist all suggest that a sophisticated analysis program would be appropriate and advantageous. Acknowledging that present methods yield satisfactory, albeit conservative, results, one of the primary goals of this research project is to provide an alternate, improved method of analysis that uses the finite element method (FEM).

The finite element method is an accepted way of analyzing structures. Specialization of an FEM code for a particular problem can lead to economy of design. Currently there exists a unique finite element program created solely to analyze prestressed concrete slabs. NOPARC (NONlinear Analysis of Prestressed AND Reinforced Concrete Slabs and Panels) is a finite element program developed at the University of California, Berkeley, to:

[T]race the quasi-static response of reinforced and prestressed concrete slabs and panels under short time and sustained load conditions. Time-dependent effects due to load history, temperature, creep, shrinkage and aging of concrete and stress relaxation in prestressing steel are incorporated. The load-deflection history of such structures through the elastic, inelastic and ultimate ranges is calculated (van Greunen 1979).

Operation and specifics of this program are given by van Greunen (1979).

This report accompanies the software package *TEXSLAB* which provides a user-oriented interface for the analysis program NOPARC. The software package *TEXSLAB* has been developed for use by TxDOT. Three additional reports support the accuracy of NOPARC as a viable analysis tool for flat slab structures (Roschke and Inoue 1990; Roschke, Pruski, and Smith 1992; Roschke, Pruski, and Sripadanna 1992).

2. INTRODUCTION

Because of the complexity of the finite element analysis program NOPARC, a user-oriented interface has been developed. This report details the procedure required to use *TEXSLAB* to perform an analysis using NOPARC. Even though *TEXSLAB* does not require the user to be an FEM specialist, some basics must be known. Many good references and textbooks are available. The NOPARC reference manual (van Greunen 1979) should accompany this report and contains some basic finite element theory. This report is written for a structural design engineer. Understanding of prestressed and reinforced concrete design is required. Certain restrictions exist for the use of *TEXSLAB*.

TEXSLAB can be used for analysis of many types of slab structures. Here, bridges are the focus of attention, but any four-sided solid slab can be analyzed. Slabs with varying thicknesses can be treated, but numerical simulations of these types of structures have not been experimentally verified. Slabs with voids, i.e. waffle slabs, cannot be analyzed with *TEXSLAB*. A major limitation of the analysis capabilities of NOPARC is that shear deformation is not taken into account. Thus, if the structure is shear critical, predictions generated by this program should be discredited. It is best not to use *TEXSLAB* for any problem where shear is a major concern.

This report is written as a user's manual for the software package *TEXSLAB*. Instructions on what is required for operation, program installation instructions, and general program operation are discussed. Program operation is divided into four main parts: the file editor, the input file generator, the analysis program, and the graphical viewing option. Primary operation of *TEXSLAB* occurs in IBM's *Operating System/2* TM (OS/2) (1988, 1991) on a microcomputer. The graphical viewing option requires availability of the commercial software package SuperView. Each part is discussed in detail in what follows.

3. SETTING UP *TEXSLAB*

3.1 Hardware and Software Requirements

3.1.1 Operating System

TEXSLAB operates on an IBM PC-compatible computer. IBM's *Operating System/2* (OS/2) (1988, 1991) must be installed. Version 1.2 or higher is recommended, but older versions operate satisfactorily. Minimal knowledge of the OS/2 operating system is required for proper execution of the program. See Appendix I for some useful commands and procedures used in OS/2.

3.1.2 Extended Memory

Extended memory is recommended because it can be accessed directly by OS/2. NOPARC requires considerable random access memory (RAM) and access to temporary storage space to perform the analysis. When available, extended memory is used by the code; otherwise, hard disk space is utilized as physical memory. Physical memory, which is not actually present but is deemed to exist because of the fixed disk, is termed virtual memory. OS/2 creates virtual memory automatically as the need arises. A swap file is a file that occupies hard disk space that the computer uses for allocation of additional RAM memory. The swap file is always present but increases in size when the amount of RAM memory required exceeds available extended memory. Sufficient disk space must be present to accommodate the increase in swap file size that results when extended memory is not adequate. The location of the swap file is specified in the `CONFIG.SYS` file. Details are explained in the OS/2 manual. Program execution is slower when virtual memory is used than when physical extended memory is available. Disk space required for the swap file is in addition to space necessary for program installation and execution as explained in the following sections.

3.1.3 Hard Disk Space

NOPARC reads and writes temporary and permanent files to and from the disk. The magnitude of these files is such that a hard disk is required. The amount of hard disk space required depends on the size of model to be analyzed and the RAM memory allocation as previously discussed. For a user who is learning the software, 10.0 MB of disk space is sufficient to perform a complete analysis on a

small sample problem. The program itself occupies 1.5 MB of disk space. After the installation procedure is complete, *TEXSLAB* can be run without performing an analysis with 1.0 MB of available space.

3.2 Installation

To install *TEXSLAB*, copy all files from the two program diskettes to directories on a hard disk. This can be done in either the OS/2 or DOS operating system by using the following commands, where it is assumed that A is the floppy drive and C is the hard disk drive:

```
C:\>MD TEXSLAB
C:\>COPY A:*. * C:\TEXSLAB.
```

An additional directory is required for graphics files and is set up as follows:

```
C:\>MD ALGOR
C:\>COPY A:\ALGOR\*. * C:\ALGOR.
```

Note that the ALGOR directory must be in the root directory of the drive where *TEXSLAB* is installed. ALGOR library files also need to be copied to the hard drive. From within the ALGOR directory, the following commands can be used:

```
C:\ALGOR>MD OVL
C:\>COPY A:\ALGOR\OVL\*. * C:\ALGOR\OVL.
```

The following files are required to run *TEXSLAB*. Location and prospective operating system for execution are specified. In addition, data and library files are required and specified correspondingly.

Disk 1

Screen Editor Files

C:\TEXSLAB\TEXSLAB.EXE	<OS/2>
C:\TEXSLAB\MENU.EXE	<OS/2>
C:\TEXSLAB\MAKMICI.EXE	<OS/2>
C:\TEXSLAB\MAKMICO.EXE	<OS/2>
C:\TEXSLAB\MENU1.MNU	<DATA>
C:\TEXSLAB\MENU2.MNU	<DATA>
C:\TEXSLAB\MENU3.MNU	<DATA>

C:\TEXSLAB\MENU4.MNU <DATA>
 C:\TEXSLAB\MENU5.MNU <DATA>
 C:\TEXSLAB\MENU6.MNU <DATA>
 C:\TEXSLAB\DEFAULT.IFL <DATA>
 C:\TEXSLAB\DEFAULT.OFL <DATA>

Transform Option File

C:\TEXSLAB\MICGEN.EXE <OS/2>

Analysis File

C:\TEXSLAB\NOPARC.EXE <OS/2>

Disk 2

View Option Files

C:\TEXSLAB\STARTSV.EXE <DOS>
 C:\TEXSLAB\MKSV.EXE <DOS>
 *C:\TEXSLAB\SVIEW.EXE <DOS>
 C:\ALGOR\SETGRAPH.EXE <DOS>
 C:\ALGOR\SETMODE.EXE <DOS>
 C:\ALGOR\SETGRAPH.DAT <DATA>
 C:\ALGOR\OVL*.OVL <LIBRARY>

Sample Exercise Files

C:\TEXSLAB\PROFILE <DATA>
 C:\TEXSLAB\SAMPLE <DATA>
 C:\TEXSLAB\SAMPLE.DAT <DATA>
 C:\TEXSLAB\SAMPLE.DO <DATA>
 C:\TEXSLAB\SAMPLE.IFL <DATA>
 C:\TEXSLAB\SAMPLE.OFL <DATA>
 C:\TEXSLAB\SAMPLE.OUT <DATA>
 C:\TEXSLAB\SAMPLE.NSO <DATA>
 C:\TEXSLAB\SAMPLE.SST <GRAPHICS>
 C:\TEXSLAB\SAMPLE.SVD <GRAPHICS>

Notes:

*For proper execution of SVIEW.EXE, an ALGOR configuration file must exist on drive C in the ALGOR directory. Follow the procedure outlined in steps 1 through

7 to create the file *ALGOR.CFG*. This procedure is performed only at installation of the program or when monitor or input device have been changed.

1. Get into the DOS operating system.
2. Change to the *ALGOR* directory.
C:\>CD ALGOR
3. Execute *SETGRAPH* by using the command
C:\ALGOR\>SETGRAPH
4. Respond to inquires about monitor type and input device.
5. Exit to *SETMODE* when complete.
6. Select monitor resolution.
7. Exit.

****Multiple library files exist and accompany software to suit a variety of user hardware configurations.**

3.3 Configuration

For *TEXSLAB* to execute SuperView, the *AUTOEXEC.BAT* file must have a *PATH* environmental command that places the *ALGOR* directory in the search path. A text editor can be used on the *AUTOEXEC.BAT* file for this purpose.

3.4 Execution

After all files have been installed, OS/2 Presentation Manager needs to be set up to control program execution. Appendix I details the procedure suggested for setting up the presentation manager to control *TEXSLAB*. Depending on what version of OS/2 is installed, do the following to begin the program.

Version 1.1 of OS/2

Manually start *TEXSLAB* from the keyboard at the prompt:

[C:\TEXSLAB]TEXSLAB.

Version 1.2 or higher of OS/2

1. From Desktop Manager icons, select *TEXSLAB*.
2. Select the *TEXSLAB* icon from the *TEXSLAB* group to begin generation of the input file.

4. DESCRIPTION OF PROGRAM

TEXSLAB uses a menu structure to access and control execution. This chapter is divided into five sections. Each section corresponds to one of the major menu items that are displayed at the uppermost line of the main screen of *TEXSLAB*. These items and their general functions are as follows:

- **FILE:** Commands for handling data files.
- **EDIT:** Commands for creating model data and controlling output.
- **TRANSFORM:** Command to execute input model generation.
- **VIEW:** Commands for graphical display of model and results.
- **ANALYZE:** Command to execute analysis program.

These major subdivisions of the program are integrated under a master menu structure which allows the user easy access to various capabilities of the program. The name of the current input file as well as the status with respect to input generation and analysis are also displayed.

These five modules and their subdivisions are described in the following sections. Options are selected from the main menu by using the direction arrow keys to highlight a desired option, and then pressing the [Enter] key. Procedures for selecting an option from a pull-down menu are the same as for the main menu except that the Up and Down arrow keys ($\uparrow\downarrow$) are used to move from one option to another. Using the Left and Right arrow keys ($\leftarrow \rightarrow$) within a pull-down menu allows movement from one menu to the next. The option highlighted within this new pull-down menu is the most recently selected option. The [Esc] key can be used from within a submenu to return to the menu at the next higher level. Instructions or messages regarding a main or submenu option are displayed at the bottom of the screen where the line immediately above the bottom line contains an error message or message relating to the action carried out.

In the description that follows, selection of an option implies use of this scheme without specific mention.

4.1 File Menu

In order to perform an analysis with *TEXSLAB*, an input file which describes geometry, materials, loads, and prestressing information must be prepared. The name for this file must follow naming conventions specified by DOS. The FILE

module prompts the user to specify a filename and facilitates use of file and directory utilities. A point to be noted is that all filenames are entered without any extension and can be no longer than eight characters. The specified filename is assigned numerous extension names as explained in chapter 8.

4.1.1 New

To create a new data file select the option **New**. The user requests a new input file by entering a filename which does not exist in the current default directory. When a new filename is specified, the existing default file is duplicated and renamed according to the name requested. If the specified filename already exists in the directory, a message is displayed that prompts the user for a new filename.

4.1.2 Open

Select the **Open** option when edits are to be performed on an existing input file. The filename must match an existing filename in the directory where the program is operating. File format must correspond to the way it was created originally by the program. For example, if the file was edited outside of *TEXSLAB* with a generic text editor, errors may be encountered when the program editor reads the file.

Files that have been opened or are new are closed if another file is requested for editing. If **New** or **Open** is selected after a file has been successfully started, *TEXSLAB* prompts the user if the current file is to be closed. Type "y" or "Y" if a different file is to be edited or "n" or "N" if not finished with the current file.

4.1.3 Change Name

To start a new file that makes use of an existing file, choose **Change Name** after the existing file has been opened using the **Open** command. When selected, enter the new filename to be given to the old file. Doing this enables the user to assign a different default file to be used. Use of this command saves editing time when many of the input parameters are duplicates of those in an existing file. This operation resembles the DOS copy command.

4.1.4 Directory

When **Directory** is selected, the names of all input files for *TEXSLAB* (also known as microfiles) in the current working directory are displayed.

4.1.5 Exit

To terminate operation of *TEXSLAB*, choose **Exit** from the **File** menu. Data are automatically saved if the file has been edited when the user escapes (pushes **Esc** several times) out of the **Edit** menu.

4.2 Edit Menu

The **Edit** module provides for creation of new input data files or modification of existing data files. The files created using **Edit** are called microfiles. Microfiles are small input data files used in creation of the large input data file necessary for analysis by NOPARC. Two entries exist in this submenu: **Bridge Input** and **Bridge Output**. Each option allows the user to input and/or edit basic data for a problem. **Bridge Input** supplies information required for generation of the analysis input file. **Bridge Output** specifies the kinds of output that are desired from the analysis.

Default data exist for fields in both the input and output menus. The default values are changed when the user edits fields within the popup menu structure. The user can easily move between menus and fields on the screen by using the **up** and **down** cursor keys. The current field is highlighted. An existing value may be modified or deleted using the **delete** key on the keyboard. Each character in the entry is deleted separately by pressing this key. All real numbers must contain a decimal point. **Left** and **right** cursor keys can be used to move around within a field. New characters typed are added to the existing value in the field. When the user presses **[Enter]**, the value in the buffer is written to the current field. If the user presses the **up** or **down** cursor keys, the value displayed in the highlighted area is written to the current field, and the highlight is moved from the current field to a new field, depending on which direction was previously indicated by the cursor key. If at any time while the user is typing data into the buffer, the escape **[Esc]** key is pressed, the buffer is cleared and no changes are made to data on the screen. After all data are entered for a given menu, the user must select **Done** for the newly entered data to be saved. Selecting **Done** does not save the data, but instructs the program that the newly entered data are to be saved. To save the data to the file,

press [Esc] several times until the cursor rests in the menu box of **Bridge Input** and **Bridge Output**. At this point, the data are saved.

4.3 Transform Menu

Transform takes the user's microfiles that are created within the **Edit** menu and converts them into an analysis input file and two SuperView input files. SuperView files are used for graphical representation of the input model as discussed in section 4.4. The analysis input file is read in when **Run** is selected from the ANALYZE menu (see section 4.5). This item is not available unless the user has created a microfile for input. A default microfile for output parameters is used if one is not created. Transforming the microfile into an input file for analysis can require several minutes or longer depending on the model size. A status box in the lower right hand corner of the screen notifies the user when the transformation process is complete. Do not edit a new file or do anything else in the *TEXSLAB* program while a microfile is being transformed.

4.4 View Menu

4.4.1 Preview

Preview allows the user to graphically view the finite element mesh, support locations, prestressing tendon configuration, and what and when the loads are applied before running the finite element analysis. SuperView is used to display the model graphically. When **Preview** is selected, the user is prompted to either select **Plan View** or **Profile View**. Select **Plan View** to show the basic geometry of the bridge, discretization of the model, support locations, prestressing tendon locations and applied forces, and the applied loads. Select **Profile View** to view a longitudinal cross-section of the bridge that shows the slab thickness, location of reinforcement, and a single tendon profile. The first longitudinal tendon of a group with more than two inflection points is displayed or, if the profile is straight (i.e. there are only two inflection points), the first tendon of the first tendon group is displayed.

SuperView only operates in DOS. To switch to the DOS mode, press [Ctrl]+[Esc] to move to the OS/2 Task Manager window. From here, select **Group - TEXSLAB** and then **ALGOR SuperView** from the *TEXSLAB* group. With

the mouse, click twice on **ALGOR SuperView** to view the model. Before SuperView can be run, the file **MKSV.EXE** must be run to convert the ASCII file created by **Transform** into a binary file. Thus, **MKSV.EXE** runs first; SuperView follows by displaying the newly created finite element model on screen.

Appendix II contains useful SuperView commands for checking various model parameters. After checking the model, exit out of SuperView by clicking on the **quit** option. The user is returned to the Desktop Manager screen. Double click on the *TEXSLAB* icon at the bottom of the screen to get back to the main menu screen. **DO NOT** click on the OS/2 icon in the *TEXSLAB* group.

If an error was found in the model with SuperView, a correction can be easily made by going back to the **EDIT** menu and modifying the input. After modification is made, redo the **Preview** procedure to display the corrected model.

SuperView cannot be used to check all bridge input. It is recommended that all menu items be double checked to insure proper modeling.

4.4.2 Postview

Postview can be selected after analysis has been performed. Analysis results are examined graphically by using SuperView. Upon selection of **Postview**, the user is prompted to make a selection of the concrete layer in which the stress is to be displayed. Because SuperView requires separate files for the stresses at each layer, the user must select each layer separately. When one of the layers is selected, the file containing that layer's stress values is copied to the file used by SuperView. After that layer's stress has been examined, the user is required to get back to the *TEXSLAB* program to select the next layer so that a file manipulation can take place and the correct file is used in SuperView. Appendix II contains useful SuperView commands to aid the user in presentation of results.

Results displayed graphically are not always identical with those predicted by the analysis. Displacements that are displayed are identical to computed values because SuperView uses the analysis values directly. However, stresses are manipulated to suit the graphical file format that is different from the format of the analysis output. The analysis provides values at three integration points. SuperView requires stress values at the nodes of each element. Conversion of element stresses occurring at the integration points to stresses at the nodes takes place in the analysis program. Thus, if the user compares conventional text output from the analysis to graphical output, there may be a slight discrepancy.

Whether graphical output is available and what it is to represent are controlled with the **Bridge Output** submenu that is located in the EDIT module. Since multiple time and load analyses can be performed in one run, the user specifies graphical analyses for which results are to be output. By default the program prints output results for all analyses. If output is not provided for all analyses, the load configuration shown on the screen will not necessarily correspond to the results displayed on the screen.

4.5 Analyze Menu

ANALYZE controls execution of the NOPARC analysis. Two choices exist in this menu option. Each choice is explained in the following sections.

4.5.1 Run

Selection of **Run** triggers execution of the analysis code NOPARC. As explained previously, **Transform** supplies the input data file that NOPARC reads. Thus, the microfile must be transformed prior to selection of **Run** from the ANALYZE menu. The FEM analysis requires a great deal of disk space and takes a considerable amount of time. Table 1 gives some examples as to how much time and disk space are required for a variety of model sizes. An IBM PS/2 Model 80 computer was used to obtain the results. This is a computer that has an Intel 80386 processor running at a clock speed of 25 megahertz, a math coprocessor, and eight megabytes of RAM.

TABLE 1. Examples of Disk Space and Execution Times

Case (1)	Number of Nodes (2)	Number of Elements (3)	Number of Tendons (4)	Number of Analyses (5)	Diskspace (Mbytes)* (6)	Time (min) (7)
1	637	1152	99	3	55-60	200
2	637	1152	99	9	55-65	540
3	527	960	47	8	45-50	360
4	306	528	47	1	23-24	36
5	198	336	38	4	13-15	72

* Disk space required depends on quantities requested for output.

When an analysis is taking place, no additional file opening or editing may take place within *TEXSLAB*. *TEXSLAB* may be left running in the background, and other applications may be executed in the foreground.

4.5.2 Kill

Selection of **Kill** terminates the NOPARC analysis program. Doing this leaves large scratch files on the hard drive. If **Kill** is selected, exit from *TEXSLAB* and delete the files that start with ZZ. Also, delete the output data file associated with the analysis.

5. INPUT EDITOR

Within the EDIT menu two options exist: **Bridge Input** and **Bridge Output**. Select **Bridge Input** to create a bridge model. Upon selection of **Bridge Input**, a menu with seven options pops up on the screen. Each of these options controls different aspects of model generation and must be accessed. The order in which each is selected is important. Generally, starting with the first option, **Geometric Parameters**, and working through the list in a downward fashion is recommended.

A sign convention exists for *TEXSLAB*. When material properties are required, input positive values for both compressive and tensile strengths as specified. For thicknesses and eccentricities, specify positive values for distances above a reference plane and negative values for distances below the plane.

In many menus, default data are provided that often can be used for the new problem. Default options for data are not provided where specific geometry for the given problem is required. All default values may be changed by the user, if desired. To make changes, activate the individual cell, delete the existing data, type in new values, and press [Enter].

5.1 Geometric Parameters

Select geometric parameters to begin the input. A submenu with seven options appears. Each option is discussed in detail below.

1. **CORNER POINTS OF THE SLAB** - This submenu selection refers to geometric boundaries of the bridge slab. Coordinates are input starting with the lower left corner of the bridge and continuing counterclockwise. The points must be input in a counterclockwise order for proper file generation. Fig. 2 shows an example slab that is to be modeled. Corner points are labeled. A description of the input for corner 1 is given below. The global X- and Y-coordinates are shown on Cartesian coordinate axes at the lower left corner of the model.

- a. **Bottom left corner X-coordinate:** Enter global X-coordinate for corner one.
- b. **Y-coordinate:** Enter global Y-coordinate for corner one.

Coordinates of the remaining three corner points are input in the same fashion.

2. **COLUMN CONTROL PARAMETERS** - Select this option to input information about column supports and transverse mesh divisions.

- a. **Number of column lines:** This option refers to the number of interior lines of support in the transverse direction that are present. Interior supports are primarily columns. Specification of the number of column lines dictates the number of spans. The number of spans equals one plus the number of column lines. A limit of nine column lines exists. There are one column line and two spans in Fig. 2.
- b. **Number of columns in a line:** This option refers to the number of columns in a single line. All lines are assumed to have the same number of columns. Up to nine columns per column line can occur. There are four columns in the line shown in Fig. 2

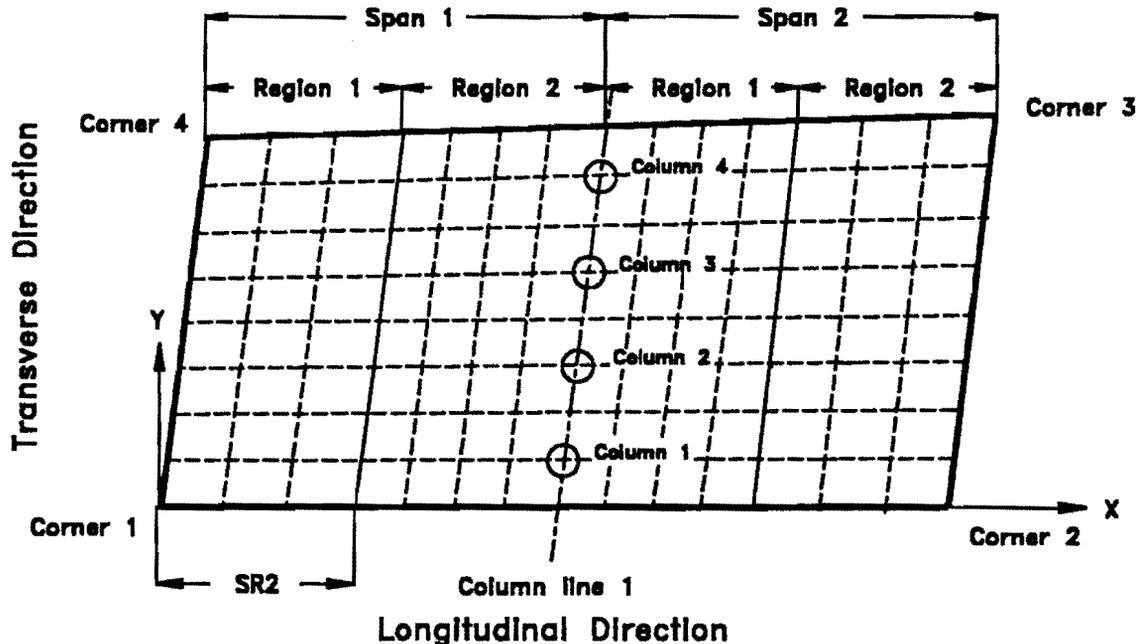


FIG. 2. Geometric Representation of Input Model

- c. **Number of mesh divisions transversely:** This number depends on how many columns are in a line and how coarse the mesh will be in the transverse direction. (Column locations are input at a later stage.) Consideration of column spacing and placement is necessary when specifying this number so that sufficient accuracy in modeling is possible. Column supports can only occur at nodes of the finite elements. An arrangement such that nodal coordinates match column coordinates is sought. Transverse mesh divisions are uniform across the width of the model. The dimension of each division is determined by dividing the width of the model at the cross-section of interest by

the number of transverse divisions specified. Irregular column spacing cannot occur unless an impractical number of transverse divisions are specified. Fineness of mesh is also controlled with this number. Consideration of lane loading may also be important, as discussed later. No more than forty transverse mesh divisions may be specified. For Fig. 2 the number of divisions in the transverse direction is eight.

3. **COORDINATES OF COLUMN LINES** - If columns exist, this option should be selected next. Coordinates for each column in each line are required. Select each line independently from within this submenu. Enter global X- and Y-coordinates for each column. The input values may not correspond to the values generated for the input data file. The node closest to the location where the column is specified becomes the actual location for the column in the analysis. A straight line drawn between the coordinates of the first and last columns in a line indicates the end of one span and the start of the next span. As an example, input for the first column in column line one is given below (Fig. 2).

- a. **X-coordinate - column 1:** Enter global X-coordinate for column one in column line one.
- b. **Y-coordinate:** Enter global Y-coordinate for column one in column line one.

4. **GENERAL SPAN INFORMATION** - Select this option to specify how many individual regions exist within each span. Regions refer to major subdivisions within a span. The number of regions may be set so that different layer properties can be assigned or different degrees of mesh refinement can be established. If the slab has a uniform thickness and a uniform number of longitudinal mesh divisions, one region per span is recommended. For a slab with a variable thickness along the span, multiple regions should be selected so that variation can be specified. A limit of ten regions per span is set. For example, there are two regions in span one of Fig. 2. The special case of modeling a variable depth slab bridge is depicted in Fig. 3. This problem can be solved by assigning multiple layer systems to different regions in the model. The input of this problem is not discussed here, but knowledge of the existence of such type of structure is important and worth noting.

5. **SPAN SPECIFIC INFORMATION** - This menu allows specification of parameters within each region of each span. When this option is selected, a span

number must be chosen. The span selected requires information for each region as specified in the previous menu. If there is only one region, the number of mesh divisions longitudinally is specified for the entire span. When multiple regions exist for a span, information on longitudinal divisions and the start of each region is required. The start of a region is given by specifying the global X distance from the start of the span along the bottom (Line 1-2) to the beginning of the region specified. The first region of every span always starts at the beginning of the span; thus, the distance from the start of the span is zero, which is trivial and not included as input. The sum of the number of divisions for each region in a single span determines the number of division for that span. Each region is assigned a concrete layer system number and a reinforcing steel layer system number as discussed in what follows. Region numbers start with one at the left side of the span and increase to the right. Region numbers begin again with one when a new span is encountered. All regions for each span require specification of these parameters. Description of input parameters for the first span of Fig. 2 is given below.

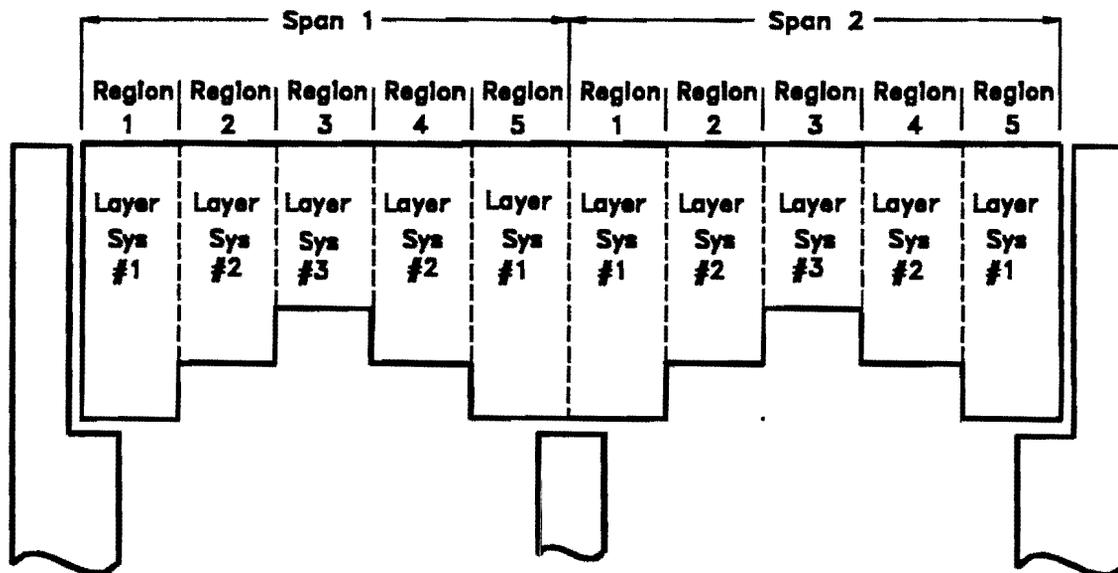


FIG. 3. Haunched Bridge Slab

- Number of divisions in region 1:** Enter number of mesh divisions in the longitudinal direction for region 1. From Fig. 2 there are three divisions separated by dashed lines.
- Location for start of region 2:** Specify the distance to the start of region 2 from the start of span 1. In Fig. 2 this distance is shown by the expression SR2.

- c. **Number of divisions in region 2:** Enter the number of mesh divisions in the longitudinal direction for region 2. There are four divisions in region 2 (Fig. 2).

6. **SPRING SUPPORT PARAMETERS** - Spring supports are required input quantities that are used to imitate reactions imposed on the slab by columns and abutments. Specification of spring stiffness in the vertical, longitudinal, and transverse direction is required at column and abutment locations. Springs must be specified if reaction forces are needed as output. The user can closely simulate a fixed support by specifying a very high stiffness. Specifying zero stiffness means that the slab is unrestrained in that direction.

- a. **Vertical spring stiffness at abutments:** Enter vertical spring stiffness of equivalent support at abutments per length of abutment.
- b. **Longitudinal spring stiffness at abutments:** Enter longitudinal spring stiffness of equivalent support at abutments per length of abutment.
- c. **Transverse spring stiffness at abutments:** Enter transverse spring stiffness of equivalent support at abutments per length of abutment.
- d. **Vertical spring stiffness at columns:** Enter vertical spring stiffness of equivalent support at columns.
- e. **Longitudinal spring stiffness at columns:** Enter longitudinal spring stiffness of equivalent support at columns.
- f. **Transverse spring stiffness at columns:** Enter transverse spring stiffness of equivalent support at columns.

5.2 Concrete

Select **Concrete** to input material properties and thickness of the bridge. Many values for concrete material properties are decided by defaults from the American Concrete Institute (ACI) (1989) within the analysis program NOPARC. Default values are assigned with respect to the 28-day compressive strength. Six submenu options appear when **Concrete** is selected. Each is discussed below.

1. **CONCRETE DATA** - Input of basic concrete material properties and concrete layer control occurs in this submenu. Also, whether or not creep and shrinkage calculations are performed is specified here.

- a. **Compressive strength:** Concrete compressive strength in pounds per square inch (psi) at 28 days is given here. Compressive strengths at other days are determined according to ACI Committee 209 (1970) as discussed on page 57 in the NOPARC reference manual (van Greunen 1979).
- b. **Tensile strength:** Tensile strength of concrete is specified or, alternatively, a value based on the compressive strength (*Prediction* 1970) can be used as explained on page 57 in the NOPARC reference manual (van Greunen 1979). Units for tensile strength are in psi. If an ACI default value is desired, specify -1.0.
- c. **Poisson's ratio:** Specify Poisson's ratio of concrete here.
- d. **Specific weight:** Enter the specific weight of the concrete.
- e. **Slump of material:** Specify the slump of the concrete in inches.
- f. **Number of concrete layer systems:** The number of concrete layer systems corresponds to how many different thicknesses the model may have. A different layer system is required for each thickness. If a uniform thickness is the case, specify 1. A maximum of 11 layer systems is allowed.
- g. **Creep analysis considered (yes/no)?:** A yes or no response is required here. Pressing the spacebar on the keyboard changes the no to yes, or yes to no. (This same procedure for selecting yes and no is used extensively in other parts of the program.) Creep is a time-dependent phenomenon; it is only considered when a time analysis is performed and yes is selected here. Creep parameters follow those recommended by ACI Committee 209 (van Greunen 1979).
- h. **Shrink analysis considered (yes/no)?:** A yes or no response is required here. Like creep, shrinkage is a time-dependent phenomenon and is only considered when a time analysis is performed and yes is selected here. Creep parameters follow those recommended by ACI Committee 209 (1970) (van Greunen 1979).

2. **CONCRETE LAYER SYSTEM CONTROL** - Within this submenu, specify the number of layers of concrete to be used within each system. The number of entries contained in this submenu is equal to the value in *d* in the previous section. For example, input for the layers in system 1 is described as follows:

Number of layers in system 1: Specify the number of concrete layers through the thickness of the slab. NOPARC allows up to ten layers in a system. At least four layers are recommended to provide reasonably

good results for stresses. If loading is such that cracking is expected in the concrete, more layers may be wanted so that crack propagation can be observed (van Greunen 1979). A sample cross-section with six layers is shown in Fig. 4.

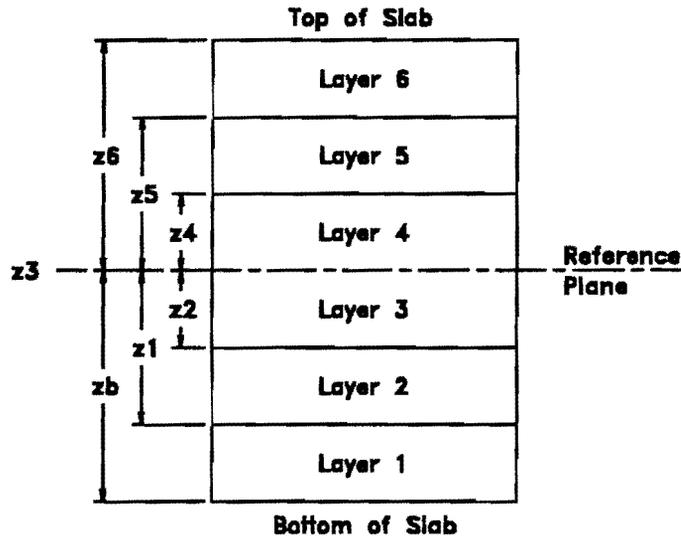


FIG. 4. Concrete Layer System

3. **CONCRETE LAYER BOUNDARY COORDINATES** - Slab thickness is set in this submenu. Boundary and intermediate layer coordinates are specified. Slab thickness is determined by adding the distance from the reference plane to the bottom of the slab to the distance from the reference plane to the top of the uppermost layer or the top of the section. Choose each concrete layer system from this submenu and input the layer coordinates in the appropriate fields. The location of the reference plane should be constant for all systems. Thus, always place an interior layer coordinate at zero.

- a. **Bottom of slab:** Distance to the bottom of the slab from the reference plane is required. This dimension, like all dimensions for this menu, is in inches. Specification of a negative number that indicates the bottom of the slab is below the reference plane is required. In Fig. 4, the variable z_b represents the distance from the reference plane to the bottom of the slab. In this case $-|z_b|$ is the required input quantity.
- b. **Lower boundary for layer #:** Start with layer one and enter the distance from the reference layer to the top of the layer specified. From Fig. 4, input $-|z_1|$. A negative value indicates the top of the layer is below the reference plane.

4. **CONCRETE LAYER REGION ASSIGNMENT** - Assign a concrete layer system to each region for each span. Refer to Fig. 3 for an example of assignment of system layers.

5. **TEMPERATURE AND HUMIDITY** - Enter the temperature and humidity at the time of placement of the concrete slab. Whether or not temperature variation is included in the analysis is also specified.

- a. **Initial temperature:** Enter the ambient temperature in degrees Fahrenheit when the bridge is cast.
- b. **Percent humidity:** Enter the percent relative humidity in percentage format when the bridge is cast.
- c. **Is temperature constant?:** Enter yes or no to specify if temperature is to remain constant when multiple time analyses are performed.

6. **TEMPERATURE GRADIENT** - When temperature is not constant, this submenu is available.

- a. **New temp. at ref. level:** Specify a temperature different from that at the initial time if analysis is to consider a thermal variation. Only a single temperature change is handled within this program; it is applied at the time of the last analysis.
- b. **Temperature gradient:** Enter temperature variation over the depth of the slab. Enter a positive number if the top of the slab is warmer than the bottom. Units for this number are in degrees Fahrenheit per inch of slab depth.

5.3 Reinforcing Steel

Select **Reinforcing Steel** to provide information pertaining to general reinforcing steel material properties and information about steel layer systems (see Fig. 5).

1. **REINFORCING STEEL PROPERTIES** - Reinforcing steel material properties are specified in this section.

- a. **Modulus of elasticity:** Enter modulus of elasticity for the reinforcing steel in unit of kips per square inch (ksi).
- b. **Yield stress F_y :** Enter the yield stress for reinforcing steel in ksi.

- c. **Modulus for strain-hardening:** Enter the modulus for strain-hardening for reinforcing steel in ksi. This value is important when an analysis for ultimate load is performed. Modulus of strain hardening is the modulus taken from the steel stress-strain curve after yielding has occurred. The default value that is displayed comes from experimental results (Roschke, Pruski, and Smith 1992).
- d. **Ultimate strain:** Enter strain of steel when failure occurs. This value is also used for ultimate analysis. A default value is provided.
- e. **Number of steel layer systems:** This number specifies the number of different reinforcing steel layer systems. Different layer systems are needed when reinforcing steel changes position along length (e.g. haunched slabs) or bar size changes along the length of the slab.

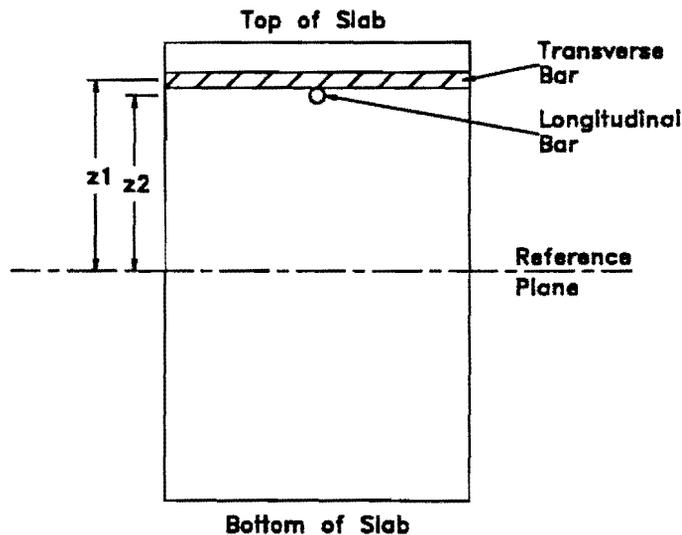


FIG. 5. Reinforcing Steel Layer System

2. **NUMBER OF REINFORCING STEEL LAYERS** - Enter the number of steel layers for each system. The number of entries contained in this submenu is equal to the value in *e* in the previous section.

Number of layers: Specify the number of reinforcing steel layers in a given system. NOPARC allows up to four layers per system.

3. **REINFORCING STEEL LAYER DATA** - A layer system number is chosen after this submenu is selected. Information for all layers is required for each system selected. One layer is discussed as an example.

- a. **Z coordinate of layer #:** Enter the distance to the centerline of the steel layer from the reference plane. If the layer is below the reference plane, enter a negative number. Distances are in inches.
- b. **Reinforcing steel number for layer #:** Enter the reinforcing steel number. For example, a number six bar corresponds to a 3/4-in. (1.91-cm) diameter bar.
- c. **Reinforcing steel spacing for layer #:** Specify center-to-center spacing of reinforcing steel for this layer. This dimension is in inches.
- d. **Angle of steel from X for layer #:** Enter the angle (in degrees) from the global X-axis that defines the direction of the reinforcing steel for this layer. For typical nonskewed rectangular geometry enter 0.0 (align with the longitudinal direction). Enter 90.0 for a transverse layer.

4. **REINFORCING STEEL LAYER REGION ASSIGNMENT** - Assign a reinforcing steel layer system number to each region for each span.

5.4 Prestressing Steel

Select **Prestressing Steel** to designate material properties for the prestressing steel. Also, system data for the prestressing steel is specified here.

1. **NUMBER OF PRESTRESSING STEEL TYPES** - Enter the number of prestressing steel systems. A system is different from another system if the area of the steel is different or any of the basic prestressing parameters change between systems. A maximum of five prestressing systems can exist.
2. **PRESTRESSING STEEL DATA** - Enter data for each system as specified above.
 - a. **Bond code for material:** Enter bond code for prestressing steel. Only one bond code may be specified for a given problem. Specify 0 for post-tensioned unbonded, 1 for post-tensioned bonded, and 2 for pretensioned steel.
 - b. **Strand area:** Enter the area of a single strand within a tendon. All strands within a tendon must have the same area. Commonly used areas for 0.5-in. (12.7-mm) and 0.6-in. (15.2-mm) strand are 0.153 in.² (100 mm²) and 0.215 in.² (139 mm²), respectively.

- c. **Number of strands in tendon:** Enter the number of strands within a tendon.
- d. **Ultimate strength of strand:** Enter the ultimate strength of the strand used for the tendon. This is given as a stress; for example, 270 ksi (1,860 MPa).
- e. **Fraction of ultimate strength:** Enter the fraction of the allowable ultimate strength for a tendon. Thus, from items *b* to *e*, the allowable force applied to the tendon is calculated as

$$\text{Initial tendon force} = \text{strand area} \times \text{number of strands} \times \\ \text{ultimate strength of strand} \times \text{fraction of ultimate strength}$$

- f. **Wobble friction coefficient:** Enter the value of the wobble friction coefficient for the group. This number is dependent on the tendon duct used. A default value of 0.0002 is for a semirigid galvanized conduit (*Post-Tensioning Manual* 1985).
- g. **Curvature friction coefficient:** Enter the curvature friction coefficient for prestressing steel. This number is dependent on the type of tendon duct used. The assigned default value is 0.25 (semirigid galvanized conduit) (*Post-Tensioning Manual* 1985).
- h. **0.1% offset yield stress:** Specify the 0.1% offset yield stress taken from the stress-strain curve for the prestressing steel material. A value of 230 ksi (1,585 MPa) is used as the default (Roschke, Pruski, and Smith 1992).
- i. **Relaxation coefficient:** Specify a relaxation coefficient for the prestressing steel. For stress-relieved prestressing steel, a value of 10 is used (van Greunen 1979). For low-relaxation prestressing steel, a value of 40 shall be used.
- j. **Number of points on stress-strain curve:** Specify the number of points used to define the stress-strain curve.

3. **STRESS-STRAIN CURVES** - Values taken from actual stress-strain curves for prestressing steel are required. Intermediate points along the curve should be specified where there is a noticeable change in slope. If stress-strain properties are not known, some default values are supplied. Default values are from actual tests performed on 270-ksi (1,860-MPa) prestressing steel strand (Roschke, Pruski, and Smith 1992). The stress-strain curve is shown in Fig. 6. A maximum of five pairs of stress-strain values can be specified.

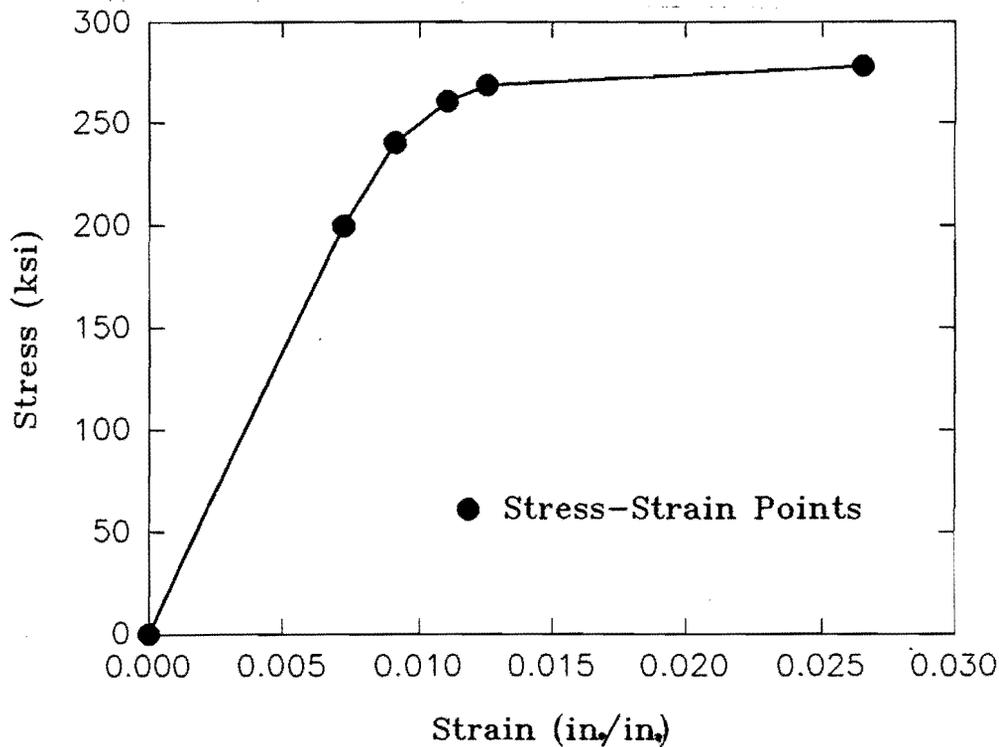


FIG. 6. Stress versus Strain for a Prestressing Tendon

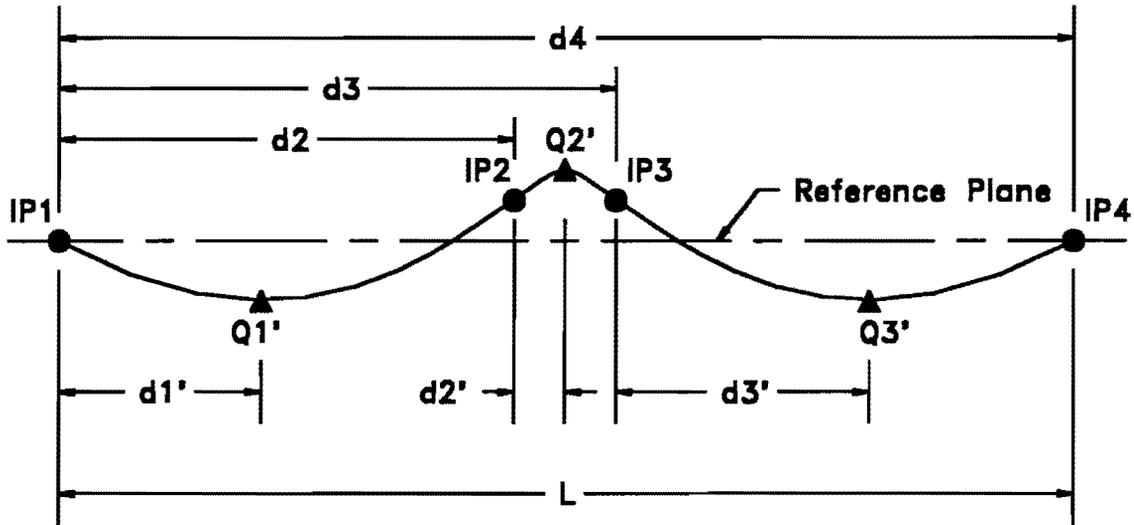
5.5 Prestressing Tendons

Data pertaining to the prestressing tendon configuration is specified within this submenu. Tendon location, prestressing force, and tendon profile are required.

1. **NUMBER OF TENDON GROUPS** - The number of tendon groups is determined by counting the number of sets of tendons that cannot be grouped together. Possible reasons for having different groups of tendons include: different prestressing steel systems are used, tendon spacing is different from group to group, tendon direction changes, or different prestressing force is specified.
2. **PRESTRESSING TENDON DATA CONTROL** - Various tendon parameters are specified within this option. All groups of tendons are created here. A single tendon group is discussed as an example.
 - a. **Number of tendons in group:** Specify the number of tendons for this group.
 - b. **Tendon material number:** Specify which prestressing steel system is to be used for this group.

- c. **Number of inflection points:** Enter the number of inflection points required to describe the tendon profile. For guidance, refer to Fig. 7, where four inflection points are shown.
- d. **Tendon direction code:** Tendon direction can be longitudinal or transverse. Enter 1 for longitudinal and 2 for transverse.
- e. **Initial tendon spacing between tendons:** Tendon spacing is required at the initial and final ends for each tendon group. Specify the distance in feet between two tendons at the starting end for this group. For group 1 in Fig. 8, S_i1 represents the tendon spacing at the initial end. For skewed edges the distance includes both the X- and Y-components.
- f. **End spacing between tendons:** Specify the distance in feet between two tendons at the final end for this group. For group 1 in Fig. 8, S_e1 represents tendon spacing at the final end.
- g. **First tendon's beginning:** Tendons can only start and end at boundaries of the bridge. Distances along the boundary from the corners of the model specify the locations of the tendons. The first tendon's beginning is the location of the first tendon of a group. For longitudinal tendons, the beginning of a tendon starts along edge 1-4. Specify the distance in feet from corner 1 to the start of the initial tendon for this group. In Fig. 8, D_i1 represents this value. For transverse tendons, the beginning of a tendon starts along edge 1-2. Specify the distance in feet from corner 1 to the starting position of the initial tendon for this group. In Fig. 8, D_i2 represents this value.
- h. **First tendon's ending:** For longitudinal tendons, the ending of a tendon occurs along edge 2-3. Specify the distance in feet from corner 2 to the end of the initial tendon for this group (D_e1). For transverse tendons, the ending of a tendon occurs along edge 4-3. Specify the distance in feet from corner 4 to the end of the initial tendon for this group (D_e2).
- i. **Anchor slip:** Anchor slip refers to the amount of set in the anchorage system after a prestressing jack releases the tendon. While a wide range of values is reported in the literature for this parameter, 0.5 in. (1.27 cm) has been chosen for the default value.
- j. **Fraction of jacking force at initial end:** Enter a fraction to be multiplied by the applied jacking force at the initial end as specified in section 5.4. Enter 1.0 to apply force or 0.0 for no force.

- k. **Fraction of jacking force at final end:** Enter a fraction to be multiplied by the applied jacking force at the final end as specified in section 5.4. Enter 1.0 to apply force or 0.0 for no force.



- Inflection Point
- ▲ Point of Maximum Eccentricity

FIG. 7. Tendon Profile

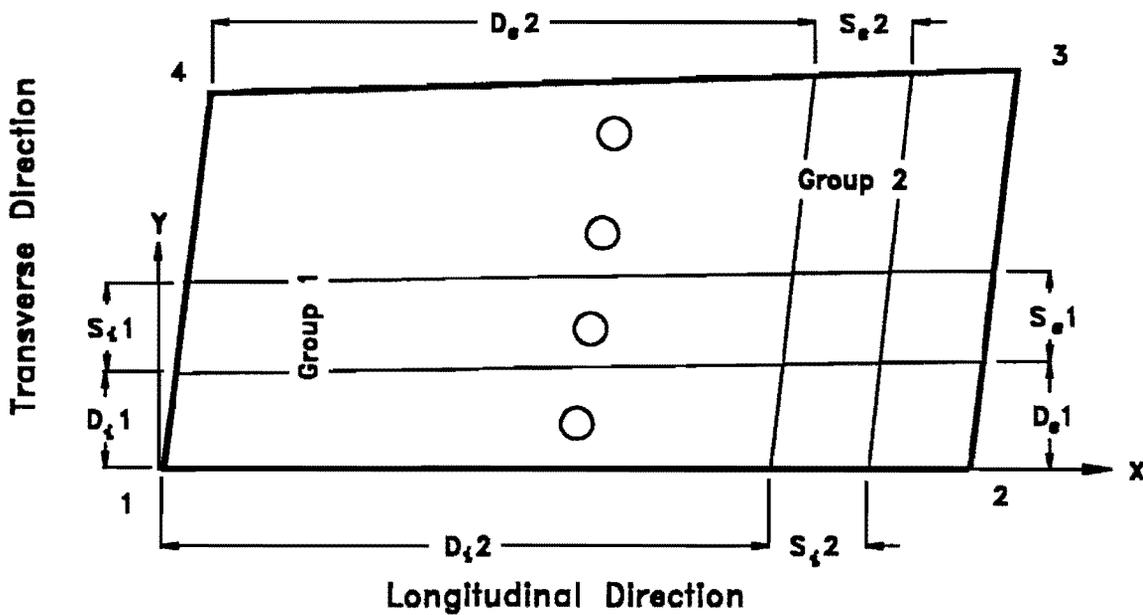


FIG. 8. Specification of Prestressing Tendon Groups

3. **INFLECTION POINT DATA** - The tendon profile is determined by specifying eccentricities at points of inflection and at points where the eccentricities are maximum. From the data that are input, straight lines or parabolic curves map the tendon profile between inflection points. Inflection point data are required for each tendon group. Description of the input data for one group is discussed below. Fig. 7 shows the profile for a sample tendon. The input of all inflection points for this tendon is explained.

Inflection point 1 (IP1)

- a. **Inflection point location (fraction):** The location for the first point of inflection must occur at the start of the tendon. The value required specifies the distance away from the start of the tendon as a fraction of the tendon length. Since this point is at the start of the tendon, a distance of zero divided by the tendon length is the value required here. Thus, for the first inflection point of any group, a value of zero is entered. The location of the first inflection point IP1 can be observed in Fig. 7.
- b. **Tendon eccentricity at the point:** Specify the distance from the reference plane to the centerline of the tendon at the inflection point. Fig. 7 shows that IP1 lies on the reference plane; thus, eccentricity is zero at this point.
- c. **Curvature code (0 or 1):** Specification of whether the tendon is straight or parabolic between inflection points is determined here. Enter 0 if the tendon is straight or 1 if the tendon has a parabolic path between inflection points. Enter 1 if the tendon resembles Fig. 7.
- d. **Distance to a point Q (fraction):** Distance does not refer to an actual distance but to a fraction of the total tendon length. Point Q is a point between the inflection point specified in part a and the next inflection point. This point is located where tendon eccentricity reaches a local maximum between the two inflection points. Point Q is required when the tendon profile is parabolic. This value is computed by dividing the distance from the preceding inflection point to point Q (d_1') by the total tendon length (L). (See Fig. 7).
- e. **Tendon eccentricity at Q:** Specify the distance from the reference plane to the centerline of the tendon at point Q.

Inflection points 2 (IP2) and 3 (IP3)

- a. **Inflection point location (fraction):** Enter the fraction of the tendon length from the start of the tendon to the internal inflection points IP2 and IP3. These values are obtained for IP2 and IP3 by dividing d_2 and d_3 , respectively, by L (Fig. 7).
- b. **Tendon eccentricity at the point:** Same as before.
- c. **Curvature code (0 or 1):** Same as before.

- d. **Distance to a point Q (fraction):** Same as before.
- e. **Tendon eccentricity at Q:** Same as before.

Inflection point 4 (IP4)

- a. **Inflection point location (fraction):** For the last inflection point, always enter 1.0. As is shown in Fig. 7, d_4 is equal to L . Therefore, d_4/L is equal to 1.0.
- b. **Tendon eccentricity at the point:** Same as before.
- c. **Curvature code (0 or 1):** Input is not required.
- d. **Distance to a point Q (fraction):** Input is not required.
- e. **Tendon eccentricity at Q:** Input is not required.

5.6 Loads

Specification of the loads that are applied to the model and the time of their application occurs within this submenu. The user should note that the position of concentrated loads does not necessarily coincide exactly with where they are applied, since they must be applied to nodes within the model. When the load is not located directly at a node, division of the load takes place. Three cases exist when the load does not fall on a node:

- i. The load is close enough to a node so that all load is applied to the node even though it does not fall directly on the node.
- ii. The load lies between two nodes. For this case the load is distributed to the two nodes. Magnitude of applied load for each node is inversely proportional to the distance to the node from the point of loading.
- iii. The load lies somewhere on a finite element but does not lie close enough to a node to fall under one of the two preceding cases. When this happens, the load is subdivided and placed on the closest four nodes in inverse proportion to the distance from each node to the point of loading.

1. **LOAD NUMBERS** - To specify the types of loads that are applied, the number of each kind of load set needs to be assigned.

- a. **Number of time steps:** Specify the number of different days an analysis is to be performed. For a time-dependent analysis to take place, a number greater than one must be specified. A maximum of eleven time analyses can be specified.
- b. **Number of lanes:** Specify the number of individual lanes of loading. Loads applied for a lane-load configuration include a region of distributed load and up to two concentrated loads. A maximum of eleven lanes of loading can be specified.

- c. **Number of trucks:** Specify the number of individual trucks to be placed on the bridge. A truck consists of six loads applied in a specified rectangular configuration. A maximum of eleven trucks can be specified.
- d. **Number of concentrated loads:** Specify the number of individual concentrated loads that are applied to the bridge. Individual concentrated loads are specified at a distinct location. If this location does not correspond to a nodal location, the load is distributed and applied to nodes in the vicinity of the specified location as explained previously. A maximum of eleven concentrated loads can be specified.
- e. **Number of distributed load regions:** Specify the number of distributed load areas applied to the model; do not include the distributed load regions specified for the lane loads. An area consists of a group of elements that are related numerically in some fashion, such as a group of elements starting with element one and ending with element ten inclusive. A maximum of eleven distributed load areas can be specified.
- f. **Number of line loads:** Specify number of line load groups. Line loads are groups of concentrated loads applied along a line. These loads are applied to nodes within a specified group. A maximum of eleven line loads can be specified.

2. **DAYS** - A time in days must be specified for each time analysis. Each day in a sequence of days must be greater than or equal to the previous day.

3. **LOAD CONTROL** - Specification of when loads are applied is controlled here. Loads, except those caused by prestressing, may be applied on any day an analysis is performed. Once a load has been applied, it may not be removed. The load exists throughout all time analyses. The load applied may be reversed by specifying an equal and opposite load at a new time.

- a. **Fraction of dead load:** Enter a number that is to be multiplied by the dead weight of the structure. Specifying 1.0 applies the total actual dead load to the structure.
- b. **Fraction of elastic deformation at transfer:** Enter the amount of elastic shortening that occurs at the time of the application of prestressing force. Prestressing always occurs on the first day specified. Elastic shortening is the deformation of the structure due to the applied prestressing loads; it results in a loss of applied force. For post-tensioned bridges, 0.5 is a reasonable estimate of this fraction (van Greunen 1979).

- c. **Load factor for lane loads:** Enter load factor to apply to lane loads for this day. Enter 0.0 if lane loads are not applied on this day.
- d. **Load factor for truck loads:** Enter load factor to apply to truck loads for this day. Enter 0.0 if truck loads are not applied on this day.
- e. **Load factor for distributed surface loads:** Enter a load factor to apply to the distributed surface loads for this day. Enter 0.0 if distributed surface loads are not applied on this day.
- f. **Load factor for concentrated loads:** Enter load factor to apply to concentrated loads for this day. Enter 0.0 if concentrated loads are not applied on this day.
- g. **Load factor for line loads:** Enter load factor to apply to line loads for this day. Enter 0.0 if line loads are not applied on this day.
- h. **Number of load steps for application of load:** Enter the number of equal divisions into which the applied load is to be divided; these divisions are applied in an equal number of load steps. Enter one if all the specified load for this analysis is to be applied at once. A number different from one and not zero dictates the number of load steps to be performed, with an even increment of the load applied to each step. If zero is entered, the load is not applied and an analysis is not performed.
- i. **Number of load steps for time-dependent analysis:** Enter the number of times the load applied due to time-dependent effects is divided by and then applied in the same number of steps. Enter one if all of the time effects for this analysis are to be applied at once. A number different from one and not zero dictates the number of load steps to be performed, with an even increment of the load applied to each step. If zero is entered, the time-dependent analysis is not made to the next time specified. If there are no additional days for time-dependent analyses to be performed, enter zero.

4. **LANE LOADS** - Application of lane loads meets current AASHTO lane loading criteria (*Standard* 1989). Up to two concentrated loads may be applied per lane and distributed load may be applied to five individual ranges per lane. Loading is determined by the user. It is based on model geometry, lane width, and the magnitude of the distributed load. (See Fig. 9.) Discussion of the input for one lane load follows. Up to eleven lane loads may be applied to the structure.

- a. **Magnitude of concentrated load:** Specify the magnitude of the concentrated loads for this lane. The load specified is distributed to the closest nodes as discussed previously.

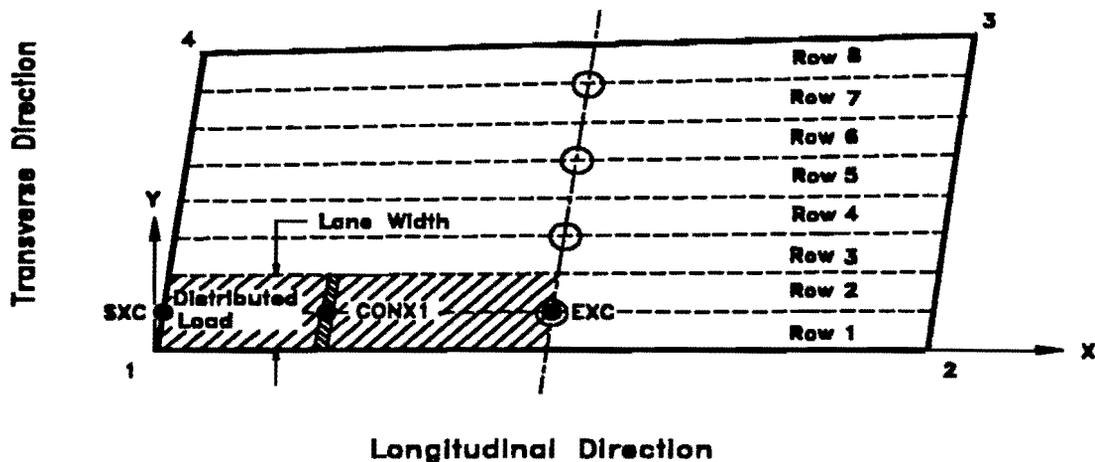


FIG. 9. Example of Lane Loading

- b. **Magnitude of distributed load:** Specify the magnitude of a distributed load that is to be placed uniformly over the width of the lane. The lane width must correspond to a multiple of element widths added together. Input for lane width is done next.
- c. **Initial element row number for lane:** Starting from the bottom boundary (line 1-2), specify the first row of elements that is loaded with a lane loading. From Fig. 9, Row 1 is the initial row of elements for the lane load.
- d. **Final element row number for lane:** Specify the last row of elements to be loaded for this lane. All rows of elements between the initial and final rows are loaded for this lane. If the initial row number equals the final row number, then only one element row is loaded. The final row number must be equal to or greater than the initial row number. From Fig. 9, Row 2 is the final row of elements for the lane load.
- e. **Concentrated load 1 x-coordinate:** Enter the global x-coordinate where the first concentrated load is applied. In Fig. 9, CONX1 represents the location where the concentrated load is applied relative to the lane.
- f. **Concentrated load 1 y-coordinate:** Enter the global y-coordinate where the first concentrated load is applied. In Fig. 9, CONX1 represents the location where the concentrated load is applied relative to the lane.
- g. **Concentrated load 2 x-coordinate:** Enter the global x-coordinate where the second concentrated load is applied. Since Fig. 9 shows positive moment loading, only one concentrated load is shown. Enter zero if the second concentrated load is not applied.
- h. **Concentrated load 2 y-coordinate:** Enter the global y-coordinate where the second concentrated load is applied. Since Fig. 9 shows positive

moment loading, only one concentrated load is shown. Enter zero if the second concentrated load is not applied.

- i. **Starting x-coordinate for range 1:** Enter the distance from end 1-4 to the start of the range for which the distributed load associated with the lane load is to begin. SXC represents this location in Fig. 9. Since SXC occurs on boundary 1-4, the value here is equal to zero.
- j. **Ending x-coordinate for range 1:** Enter the distance away from end 1-4 to the end of the lane loading for this range. EXC represents this distance (Fig. 9).

Repeat steps i and j for multiple ranges.

5. **TRUCK LOADS** - Truck loading is based on current AASHTO specifications (*Standard* 1989). Placement of a truck requires specifying the location of a single wheel and indicating the direction of travel for the truck. Placement of truck loads may not precisely correspond to the actual location of resulting loads. As discussed earlier, concentrated loads are applied at nodes. When the load does not fall on a node, it is divided among the nearest nodes. The magnitude of the wheel loads and the axle spacing are variable and are specified by the user.

- a. **Magnitude of load for front wheels:** Specify the load applied by a front wheel of the truck.
- b. **Magnitude of load for middle wheel:** Specify the load applied by a middle wheel of the truck. If the truck has only two axles, specify the load for a wheel on the rear axle.
- c. **Magnitude of load for rear wheel:** Specify the load applied by a rear wheel of the truck. If the truck has only two axles, specify zero for this value.
- d. **Truck direction:** Specify the direction of the truck. Specify zero if the truck is headed toward end 2-3 from end 1-4 and one if it is headed toward end 1-4 from end 2-3 (Fig. 10).
- e. **Right middle or rear wheel location X:** Enter the location of the right middle wheel if the truck has three axles or the location of the right rear wheel if only two axles exist. The location corresponds to the global X-coordinate (Fig. 10).
- f. **Right middle or rear wheel location Y:** Enter the location of the right middle wheel if the truck has three axles or the location of the right rear wheel if only two axles exist. The location corresponds to the global Y-coordinate (Fig. 10).
- g. **Truck width:** Specify the truck width or the distance between wheels on an axle. This defines the transverse dimension of the rectangle for which loads are applied at the boundaries.

- h. **Front truck axle spacing:** Specify the distance between the front and middle axles or the front and rear axles when only two axles exist.
 - i. **Rear truck axle spacing:** Specify the distance between the middle and rear axles. Addition of front axle spacing and rear axle spacing gives the length of the truck.
6. **CONCENTRATED LOADS** - Specification of individual concentrated loads occurs within this submenu. The location and magnitude of the load are required. Out-of-plane loading and moments about in-plane axes can be specified.
- a. **Location X:** Enter the global X-coordinate for this load.
 - b. **Location Y:** Enter the global Y-coordinate for this load.
 - c. **Z-direction load:** Enter the load applied in the global Z direction. The load applied in the negative Z direction is positive.
 - d. **Moment about X-axis:** Enter the moment applied about the global X-axis.
 - e. **Moment about Y-axis:** Enter the moment applied about the global Y-axis.

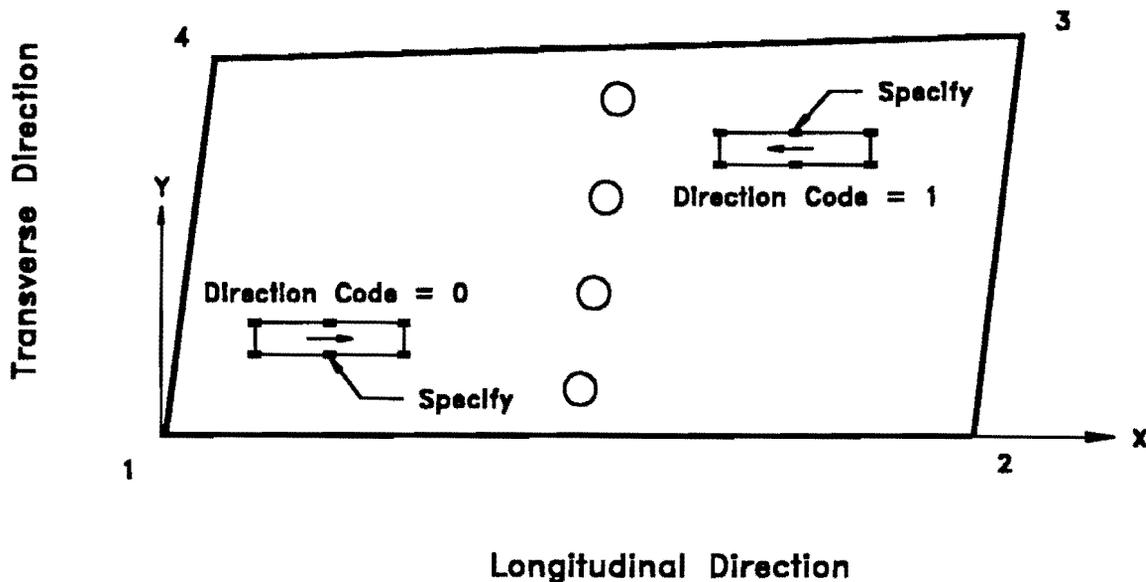


FIG. 10. Truck Loading

7. **DISTRIBUTED LOADS** - Distributed loads may be applied independently from lane loading. Application of distributed load can be specified by entering a group of elements that are to be loaded.

- a. **Magnitude of distributed load:** Specify the magnitude of the distributed load that is applied to the face of the elements specified next.

- b. **Starting element:** Enter the starting element of the group that the distributed load is applied to.
- c. **Ending element:** Enter the last element of the group that the distributed load is applied to. The ending element number must be greater than or equal to the starting element number.
- d. **Element increment:** Enter the increment that specifies the next element within the group. Specifying one gives all elements between starting and ending element numbers.

8. **LINE LOADS** - Line loads are groups of concentrated loads that are applied directly to a group of nodes. Specification of which nodes are in a group and what the magnitude of the load is for each group is required. Since node numbers are not known initially, the model should be generated and viewed before loads are applied. Viewing the model in **Preview** allows the user to specify the correct nodes for the load to be applied.

- a. **Magnitude of line load:** Specify the magnitude of the line load in units of load per length (k/ft). Loading is applied to the nodes in direct proportion to the distance between nodes. The larger the distance, the larger the load applied to the nodes.
- b. **Starting node:** Specify the starting node of the line. Node numbers increase from side 1-2 to side 4-3 and from side 1-4 to side 2-3. Thus, if the line runs longitudinally, the starting node should be the node closest to side 1-4, and if the line runs transversely, the starting node should be the node closest to side 1-2. If the spacing of nodes is equal, half as much load is applied to the starting and ending nodes as to the interior nodes (Fig. 11).
- c. **Ending node:** Specify the ending node of the line. The ending node must be in line with the starting node. "Being in line" means to be connected by a mesh line either longitudinally or transversely. Thus, a line load can either run longitudinally from side 1-4 to side 2-3 or run transversely from side 1-2 to side 4-3 (Fig. 11). A line need not start at the boundaries.

5.7 Convergence Parameters

Information input within this submenu controls convergence criteria and specifies the number of load steps per analysis. The user may not need to change any of the values within this menu from the defaults that have been established by means of extensive simulation experience (Roschke, Pruski, and Smith 1992; Roschke, Pruski, and Sripadanna 1992).

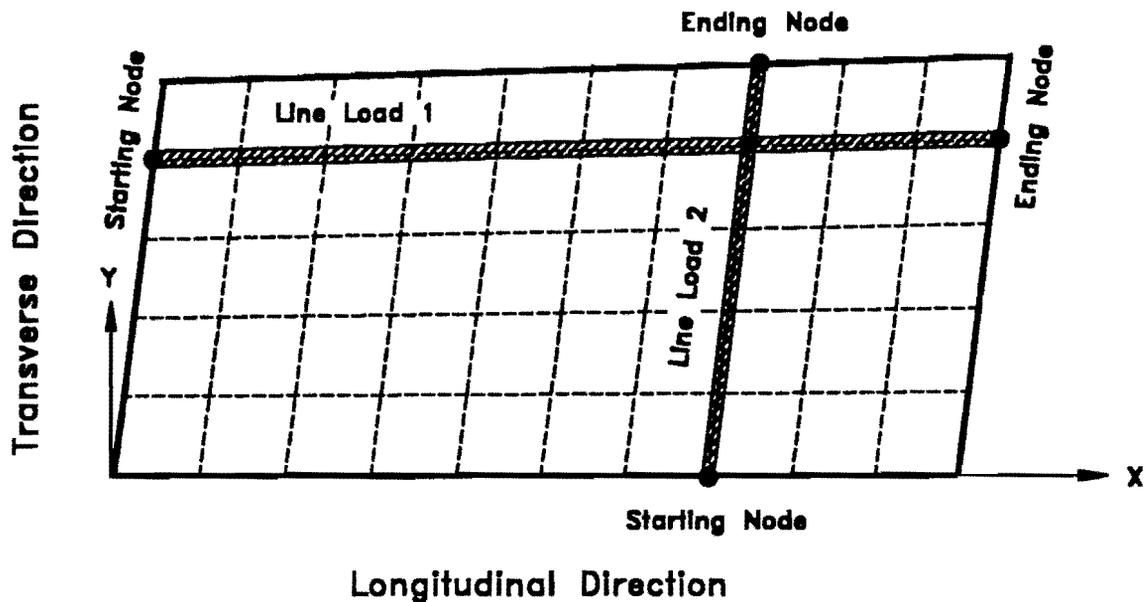


FIG. 11. Example of Line Loads

1. **CONVERGENCE CRITERIA** - Specification of the desired convergence comes from experience. A detailed explanation on convergence criteria is presented in the NOPARC reference manual (van Greunen 1979). Default values exist within *TEXSLAB*, and the user is encouraged to use them.

- a. **Displacement convergence:** Input the displacement tolerance that corresponds to the maximum component of the displacement increment for an iteration. The smaller this value is, the harder it is to obtain convergence.
- b. **Rotation convergence:** Input the rotation tolerance that corresponds to the maximum component of the rotation increment for an iteration. The smaller this value is, the harder it is to obtain convergence.

2. **UPPER LIMITS** - Specification of upper limits on displacements and rotations is required to indicate when the solution goes highly nonlinear. This usually results from very large displacements and rotations that occur after the structure fails. Values should be input here that exceed the maximum deflection and rotation expected and yet not be so large as to defeat their purpose of stopping execution of the code for excessively large displacements and rotations.

- a. **Maximum displacement:** Enter a value larger than the maximum displacement expected for any application of load that is specified.
- b. **Maximum rotation:** Enter a value larger than the maximum rotation expected for any application of load that is specified.

3. **NONLINEAR CONTROL** - Because the analysis program NOPARC performs a nonlinear analysis, an iterative method is used to obtain a solution. Within this submenu, the user specifies information related to the nonlinear analysis.

- a. **Max number of iterations for load analysis:** Enter the maximum number of iterations allowed to converge to a solution. The value entered here is for the analysis predicting the response of the model to externally applied loads. This value is related to the maximum convergence values specified in **CONVERGENCE CRITERIA**. Smaller values specified there result in more iterations being required to converge to a solution. It has been determined that slabs with design loads converge in less than ten iterations using the default convergences given.
- b. **Max number of iterations for time analysis:** Enter the maximum number of iterations allowed to converge to a solution. The value entered here is for the analysis predicting the response of the model to time-dependent loading. This value is related to the maximum convergence values specified in **CONVERGENCE CRITERIA**. Smaller values specified there result in more iterations being required to converge to a solution. Time-dependent analyses generally converge in fewer than ten iterations.
- c. **Iteration type code:** Element and global stiffness matrices can be reformed at various times. The user is given control of when the stiffness matrices are reformed. Enter a value in this field (see Table 2) to control when they are reformed within an analysis.

TABLE 2. Iteration Codes for Reformulation of Stiffness Matrices

Code (1)	Description (2)
-1	Do not reform. Use initial stiffness for entire analysis.
0	Reform stiffness for each load step.
N	Reform the stiffness each N iterations.

6. OUTPUT EDITOR

The second option in the edit menu is **Bridge Output**. Editing of entries within this menu is identical to that in the input editor. Upon selection of **Bridge Output**, a menu pops up on the screen with four options. The output that is written to an output file and the output that can be viewed graphically are controlled by these options. Each option is independent from the others, and the order of selection does not matter. All options should be selected to insure proper output.

6.1 General Control

The first branch, **General Control**, controls parameters for both alphanumeric and graphical output. Three subbranches of this selection are discussed next.

1. **NUMBER OF ANALYSES** - The user does not have control over the total number of analyses when specifying the output. The number of analyses is determined from the information the user specifies in the **Bridge Input** mode. This value may not correspond to the number of days that was specified in the **Loads** branch, as discussed in section 5.6. The total number of analyses equals the number of time-dependent analyses plus the analyses for which external loads are applied. Which analysis corresponds to which time and load is discussed in what follows. The number displayed may be edited, but the value is saved.
2. **BOUNDARY SPRING INFORMATION** - Support reactions can be viewed in the alphanumeric output file if desired. Specification can be made to output reaction data for every analysis or just at the end of all analyses.
 - a. **Output reactions for every analysis:** Select **yes** if reactions are required for all analyses. Select **no** if reactions are not wanted at all or they are wanted only at the end of all analyses.
 - b. **Output reactions at end of all analyses:** If **yes** is selected in the previous option, the entry here does not matter. However, if **no** is selected, select **yes** or **no** as is appropriate.
3. **GRAPHICAL OUTPUT** - The user has control of which analyses are available for viewing with SuperView. The most informative option is to select **yes** for all analyses. This may, in some cases, be acceptable. However, when many analyses

are required and when the model contains many elements, the size of the output files becomes very large. Therefore, accommodations have been made which allow the user to select only those analyses that are of special interest. An understanding of how NOPARC executes is required to be able to specify the correct analysis for output. This is described in what follows.

Analyses are performed for all days that are specified. In addition, excluding the first day, if any load is applied, an additional analysis is performed. Application of load is discussed in section 5.6 under **LOAD CONTROL**. Some examples of typical situations are now presented.

Case 1

Situation: One day is specified and load is applied.

Result: This is the initial analysis and only one analysis is performed.

User: Select **yes** if user wishes to view results graphically.

Case 2

Situation: Two days are specified, and additional loading is not applied on the second day.

Result: There are two analyses performed. The first is the initial day, and the second is the time-dependent analysis up to the second day.

User: Two cells are displayed prompting the user to select **yes** or **no**. The first corresponds to the initial time and the second to the time-dependent analysis to the second day.

Case 3

Situation: Two days are specified, and additional loading is applied on the second day.

Result: Three analyses are performed. The first is on the initial day, and the second is the time-dependent analysis up to the second day. A third analysis is performed because load is applied on the second day.

User: Three cells are displayed that prompt the user to select **yes** or **no**. The first corresponds to the initial time and the second to the time-dependent analysis on the second day. The third cell is for the analysis which accounts for externally applied loads on the second day.

All other cases are a combinations or extrapolations of the three cases described above. Summarizing, the first day always accounts for one analysis. Each additional day may add either one or two analyses to the total number.

6.2 Nodal Control

The second branch gives the user control of output that occurs at the nodes. Nodal quantities resulting from the analysis include displacements and forces. Only alphanumeric output is controlled here.

1. **OCCURRENCE OF OUTPUT** - Upon selection of this sub-branch, a window appears that contains a field for each analysis to be performed. The number of each analysis is determined as described in section 6.1 under **GRAPHICAL OUTPUT**. However, yes and no responses are not appropriate here and do not work. A coding system is used with the numbers 0, 1, 2, and 3 being the values to be entered in the field. A brief description of each code is given in Table 3.

TABLE 3. Description of Output Control Codes for Nodes

Code (1)	Description (2)
0	Output at the end of the last iteration for each load step for this analysis.
1	Output after each iteration in each load step for this analysis.
2	Output only at the end of all load steps and iterations for this analysis.
3	No output for this analysis.

2. **NODAL CONTROL** - This subbranch specifies the nodes for which information is to be output. There are three ways to identify these nodes:

- a. **Output data at all nodes:** Output is given at all nodes if yes is selected.
- b. **Number of individual nodes:** If no is selected for option a, individual nodes for which output is desired can be specified. The number of individual nodes that do not occur in a series (see option c) is entered here. A maximum of twenty individual nodes may be specified.
- c. **Number of nodal series:** If no was selected in option a, nodal series can be specified. A series consists of a group of nodes in which a constant increment exists between two consecutive nodes in the series. Enter the number of nodal series here. A maximum of twenty nodal series may be specified.

3. **INDIVIDUAL NODES** - This option is available only if individual nodes are to be specified. Enter the node numbers into the fields supplied when this subbranch is selected.
4. **NODAL SERIES** - This option is available only if nodal series are to be specified. Three entries are required for each series:
 - a. **Starting node:** Enter the first node of the series.
 - b. **Ending node:** Enter the last node of the series.
 - c. **Nodal increment:** Enter the increment that specifies the next node within the series. Specifying one for the increment gives all nodes between the starting and ending node numbers.
5. **EXTERNAL FORCE CONTROL** - Select this option to specify whether or not the forces that act on the nodes are to be output.
 - a. **Output external forces at nodes:** Select yes if the forces occurring at the nodes are to be output.
 - b. **Output unbalanced forces at nodes:** Select yes if the unbalanced forces occurring at the nodes are to be output.
6. **DISPLACEMENT CONTROL** - Select this option to control displacement output.
 - a. **Output displacements:** Select yes if displacements at the nodes are desired.
 - b. **Output displacements in local coordinates:** Specify yes or no to specify whether or not to output displacements in local coordinates.

6.3 Element Control

The third branch allows the user control of output from the finite elements. Element quantities include stresses and strains predicted to occur in the concrete by the FEM. Only alphanumeric output is controlled here.

1. **OCCURRENCE OF OUTPUT** - Upon selection of this subbranch, a window appears containing a field for each analysis to be performed. The number of the

analysis is determined as discussed in section 6.1 under **GRAPHICAL OUTPUT**. However, **yes** and **no** responses are not appropriate here and do not work. A coding system is used with the numbers 0, 1, 2, and 3 being the values to be entered in the field. The description of each code is described in Table 4 of section 6.2.

2. **CONCRETE LAYER OUTPUT CONTROL** - The analysis computes stresses and strains for all concrete layers specified in section 5.2. Selection of this sub-branch allows for specification of the layers for which output is desired. Three options are given to control which layers have output:

- a. **Output data for all layers of concrete:** Select **yes** if output is desired for all concrete layers.
- b. **Output data at top and bottom layers of concrete:** Select **yes** if output is desired at the outermost layers of the model. These layers represent the top and bottom of the slab.
- c. **Output data at top, bottom, and middle layers of concrete:** Select **yes** if output is wanted at the outermost layers and at the center of the model.

3. **ELEMENT CONTROL** - Specification of the elements for which output is desired occurs in this subbranch. There are three ways to identify these elements:

- a. **Output data at all elements:** Output is given for all nodes if **yes** is selected.
- b. **Number of individual elements:** If **no** is selected in option a, individual elements are specified for output. The number of individual elements that do not occur in a series is entered here. A maximum of twenty individual elements may be specified.
- c. **Number of elements series:** If **no** is selected in option a, nodal series can be specified. A series consists of a group of elements in which a constant increment exists between two consecutive elements in the series. Enter the number of element series here. A maximum of twenty element series may be specified.

4. **INDIVIDUAL ELEMENTS** - This is available only if individual elements are to be specified. Enter the element numbers into the fields supplied when this sub-branch is selected.

5. **ELEMENT SERIES** - This is available only if element series are to be specified. Three entries are required for each series:

- a. **Starting element:** Enter the first element of the series.
- b. **Ending element:** Enter the last element of the series.
- c. **Element increment:** Enter the increment that specifies the next element within the series. Specifying one for the increment gives all elements between the starting and ending element numbers.

6. **CONCRETE OUTPUT CONTROL** - Control of output for the specified elements occurs within this subbranch. Stress components computed by the analysis are given with respect to the local coordinate system of the element. The local coordinate system is forced to correspond to the global coordinate system. The sign convention for components of stress is shown in Fig. 12.

- a. **Output concrete stress:** Select yes if output for the stress components σ_x , σ_y , and τ_{xy} occurring in the concrete is desired. Stresses are output at the three integration points for the specified layers. Locations of the integration points are specified in the output.
- b. **Output concrete strains:** Select yes if the strain components ϵ_{xx} , ϵ_{yy} , and ϵ_{xy} occurring in the concrete are desired. Strains at the three integration points are averaged, and this value is output for the layers specified.

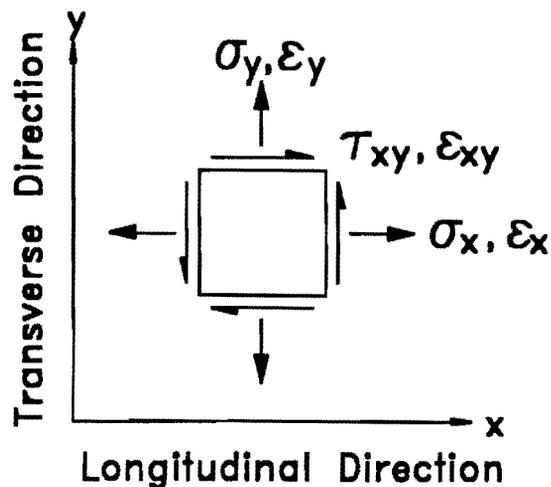


FIG. 12. Sign Convention for Components of Stress and Strain

- c. **Output principle stresses:** Select **yes** if the maximum and minimum principle stresses occurring in the concrete are desired.
- d. **Output concrete state codes:** Select **yes** if the code indicating the state of the concrete at each specified element layer is desired. Numbers that are output represent the state of the concrete. A description of these numbers is given in Table 4.

TABLE 4. Description of Output Control Codes for Elements

Code (1)	Description (2)
1	Concrete is in tension. Elastic range.
2	Concrete is in compression. Elastic range.
3	Concrete is cracked. Unloading is occurring.
4	Concrete is cracked completely.
5	Concrete is yielded in compression.
6	Concrete is crushed.

- e. **Output concrete shrinkage strains:** Select **yes** if strains due to shrinkage are to be output. Shrinkage computations are performed only when there is time-dependent analysis and shrinkage is specified in section 5.2.
- f. **Output concrete creep strains:** Select **yes** if the strains due to creep are to be output. Creep computations are performed only when there is time-dependent analysis and shrinkage is specified in section 5.2.

6.4 REINFORCEMENT/PRESTRESS CONTROL

The last branch in the output control tree allows specification of information associated with steel reinforcing bars and prestressing tendons.

1. **STEEL REINFORCEMENT** - Stresses and strains are computed by the analysis for all layers of reinforcing steel that are specified in section 5.2. The output is given in the elements of the model. Specification of the elements for which output is desired occurs in section 6.3. Output values are given at the centroid of the element.

- a. **Output steel reinforcement stresses:** Select yes if stresses occurring in the reinforcing bars within elements are to be output. These stresses are in the direction of the reinforcing bars.
- b. **Output steel reinforcement strains:** Select yes if strains occurring in the reinforcing bars within elements are to be output.

2. **PRESTRESS TENDON CONTROL** - Information that is input in sections 5.4 and 5.5 is used to compute the actual tendon profile. A detailed description of the tendon path and the force resulting at numerous locations along the tendon can be output. Also, tendon forces are calculated within the elements that are crossed by tendons for all steps in the analysis. Output of this information is specified after selection of this sub-branch.

- a. **Output prestress tendon geometry:** Select yes if alphanumeric output representing the prestressing tendons' profile and force is desired. This information is computed at the time of application of the prestressing force that occurs on the first day of the analysis.
- b. **Output information for all tendons:** Select yes if the profile of all tendons is to be output. This is not normally advisable because of the large volume of output that results. Generally, tendons within a group (see section 5.5) do not vary in profile or force. Thus, selection of a representative tendon from each group is usually sufficient.
- c. **Number of tendons to be output:** If yes is not selected in option b of this section, enter the number of tendons for which the profile is to be output. As discussed in option b, this number is often equivalent to the number of tendon groups.
- d. **Output prestress tendon forces in elements:** Select yes if the current force occurring in a tendon segment within each element is to be output. In addition, the bond code and material state code is given.

3. **TENDON SPECIFICATIONS** - Enter the individual numbers of the tendons for which profile information is to be output. Alphanumeric output is written to a file with an extension of ".out". Enter a single tendon number in each cell. The tendon numbering proceeds as follows. Tendon number one is the first tendon of the first tendon group. Tendon numbers increase by one until the total number of tendons specified for the group is reached. If more than one group exists, the first tendon of the next group corresponds to the total number of tendons contained in the previous groups plus one.

7. SAMPLE SESSION

A sample problem is presented for use as an outline for the novice user. The description that follows gives an overview of steps required to create a model, perform an analysis, and examine the results. A rectangular, three-span, slab bridge is used as a paradigm. A microfile for this example is available on the diskettes containing the *TEXSLAB* program files.

To begin, start *TEXSLAB* as described in Appendix I. Once activated, the main menu screen of *TEXSLAB* appears. This screen is used to open all menus within *TEXSLAB*. All main menus are listed in a bar at the top of the screen. To select each main menu, as required in what follows, use the arrow keys to highlight the menu that is to be selected. Press the [Enter] key to make a selection. Each of the main menus and the associated input required are discussed in what follows. As a reminder to the user, *TEXSLAB* is specifically suited for analysis of slab bridges. All menus are based on this requirement and request input accordingly. Some problems may not be well-suited to generation of the model by using *TEXSLAB*. For special problems, the user should refer to chapter 8 for an alternative method of input preparation.

7.1 File

The FILE menu must be the first selection made before an input model can be created. Since the input model has already been created (see program diskette), it exists and is now simply opened. Select **Open** and type in the name of the file without any extension; for this sample problem type "sample" and press [Enter]. File *SAMPLE.IFL* must exist in the directory */TEXSLAB/* or the directory in which this program was started from if it is not */TEXSLAB/*. As discussed in chapter 8 the file *SAMPLE.OFL* is not required to open an existing file.

7.2 Edit

Only the input microfile and the output microfile are included with the program to be used in this sample routine. Thus, for this sample session EDIT is the only other menu option that can be selected in addition to FILE. Select EDIT from the menu bar at the top of the screen. After selection, an additional submenu

box appears with two options: **Bridge Input** and **Bridge Output**. Select **Bridge Input** to view the data (previously created) that is stored in **SAMPLE.IFL** by selecting the seven options that appear on the screen. Editing of the input is not discussed here. The interested reader is directed to chapter 5, which is devoted solely to description of appropriate input in the array of branches shown on the screen. After viewing the input parameters, press [Esc] several times to get out of **Bridge Input**. Exiting **Bridge Input** automatically saves the input microfile. Return to the submenu box displaying **Bridge Input** and **Bridge Output** and select **Bridge Output**. The file **SAMPLE.OFL** is read in, and the values are ready to be viewed by selecting the four options that appear on the screen. Chapter 6 discusses each option, and the user is referred to the detailed discussion there. Upon completion of viewing the output parameters, exit again as is described above. After the input and output information has been supplied (as is now the case), the actual analysis input file can be created. This is done by selecting **TRANSFORM** from the top menu bar.

7.3 Transform

Once the microfiles have been created, select **TRANSFORM** so that they may be used to form the input file that is read in by the analysis program **NOPARC**. In addition to the analysis input file, two input files used for graphical representation of the model are created. The transform procedure may take several minutes to run. Observe the status box in the lower right corner of the screen to monitor the progress of the transformation. Do not edit a new file or do anything else in **TEXSLAB** while the status box indicates that the program is running. When transform has completed creating the new files, **Done** is displayed in the status box. Note that the file generation time is displayed in the lower left portion of the screen. Now the model can be checked graphically. Graphical representation of the model is done by the SuperView graphics package.

7.4 View

After **TRANSFORM** has run or the analysis has been completed, the model can be viewed graphically using SuperView. To invoke SuperView, the menu option **VIEW** is selected. Two options exist under **VIEW**: **Preview** and **Postview**. The availability of the two options depends on circumstances previous to selection.

Preview is the only option available after TRANSFORM has been run. The background colors for **Preview** and **Postview** are white and gray, respectively, because the analysis has not run yet and there are no new results to view. Before TRANSFORM has been run, both options are unavailable. After the analysis program NOPARC has been run, both options are available. For this sample problem, the **Preview** option is discussed first. Because ANALYZE takes considerable time, a displacement file (**SAMPLE.DO**) and a stress file (**SAMPLE.NSO**) accompany the program. Thus, even though the analysis has not run, graphical output can be viewed.

7.4.1 Preview

After the input file is created, select **Preview** to check the correctness of the model. The background color should be white. **Postview** has a gray background color because the analysis is not complete and there are no results to view. After selection, two options appear: **Plan View** and **Profile View**. Selection of each view is suggested because different parameters are checked within each option. Move the selection bar with arrow keys to select either option. Push **[Enter]** to make a selection. A box is displayed instructing the user to push **[Ctrl] + [Esc]** at the same time to invoke the OS/2 Presentation Manager. This box must be displayed on the screen before the Presentation Manager is invoked because commands are written to a work file that give instruction as to which file SuperView is to display. Within Presentation Manager, select **ALGOR-SuperView** from the *TEXSLAB* group. This causes the program **MKSV.EXE** to convert the ASCII file that is created by TRANSFORM to a binary file that is read by SuperView. After MKSV finishes, SuperView starts and the model is displayed on the screen. Check the geometry, support locations, tendon layout, and load configuration by doing the following:

Plan View

For review of the sample problem, select **Plan View** first. The plan view of the model shows a two-dimensional flat plate in the X-Y plane. Although through-thickness variation of quantities in the slab cannot be seen here, the profile view shows the through-thickness profile. After SuperView comes up on the screen, the model is displayed in some fashion. Since this is a sample procedure, a setup file **SAMPLE.SVD**, already exists which instructs SuperView how to display the model. If the view of the model is not suitable, refer to Appendix II for instructions on how

to change the view. Some general observations can be made while viewing the model on the screen or in Fig. 13. Colors on the screen are included in parenthesis.

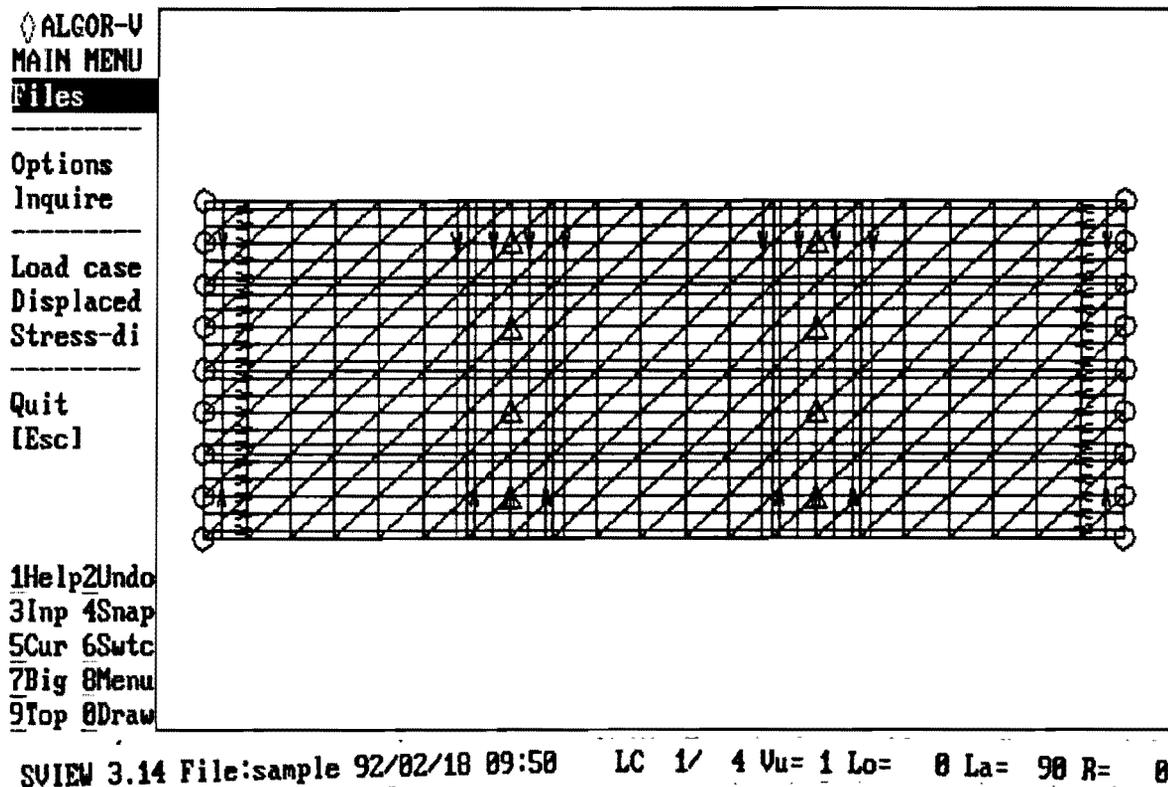


FIG. 13. Plan View of Sample Problem

- i. The model has four corners. This must always be the case when using *TEXSLAB*.
- ii. The model is rectangular. This is not a necessary condition. The model can have skew and be an arbitrary quadrilateral.
- iii. The X-axis runs horizontally across the screen. The Y-axis is oriented vertically on the screen. The origin is located at the lower left corner of the model. The X-axis is always considered to be in the longitudinal direction and the Y-axis is in the transverse direction. Any orientation of the model is allowed, and the location of the origin of the axes can be anywhere. Placing the origin at an extreme point on the model simplifies input.
- iv. The units on the screen are in inches and pounds. This corresponds to what is required for the analysis program NOPARC.
- v. The concrete slab is represented by (white) triangular plate elements. The prestressing tendons are the straight (blue) lines in Fig. 13. A

series of straight truss elements are used to represent the tendons. The reinforcing steel is not shown.

- vi. The supports are shown by means of circles and triangles. Whether the support is represented by a circle or a triangle depends on how many degrees of freedom are stiffened by springs. On the screen and in Fig. 13, columns are represented by (red) triangles and the abutments by (red) circles. See Appendix II for more information on supports.
- vii. In-plane forces are the applied prestressing force for each tendon. They are indicated by (yellow) arrows extending inward from the model's boundaries (Fig. 13).

When finished viewing the model in the plan view, select **Quit** from the **MAIN MENU** of SuperView.

Profile View

After quitting SuperView, the OS/2 Presentation Manager screen appears. Return to the *TEXSLAB* program by selecting the *TEXSLAB* icon at the bottom of the screen. Do not select **TEXSLAB** again from the *TEXSLAB* group because it is already running. After returning to the program, press [Esc] to remove the box from the screen. Push [Enter] if the selection bar is on the **VIEW** option. Select **Profile View** and activate SuperView as was done previously.

Profile View only displays the longitudinal cross-section of the model. The cross section is taken along the first longitudinal prestressing tendon. As with the plan view, this is a two-dimensional view of the slab. This time, the model is displayed in the X-Z plane. As before, a default setup file that accompanies the program instructs SuperView as to how to display the profile view. The screen should resemble Fig. 14. Some general observations are as follows:

- i. Boundaries of the concrete are represented by the straight dark black (white) lines in Fig. 14. The horizontal line that appears to divide the slab in two parts is the reference plane. Vertical lines in the interior represent division into regions. Thus, since there is one region per span, there are three spans.
- ii. The reference plane is at location $Z=0$. This should always be the case.
- iii. All Z-component values are scaled by a factor of ten.
- iv. The left edge always starts at the beginning of the prestressing tendon. On skewed structures this may be confusing because the left edge is not at $X=0$.

- v. The prestressing tendon is the multiparabolic (blue) curve.
- vi. The reinforcing steel is represented by the horizontal dashed (orange) lines.
- vii. Only the longitudinal reinforcing steel is shown.

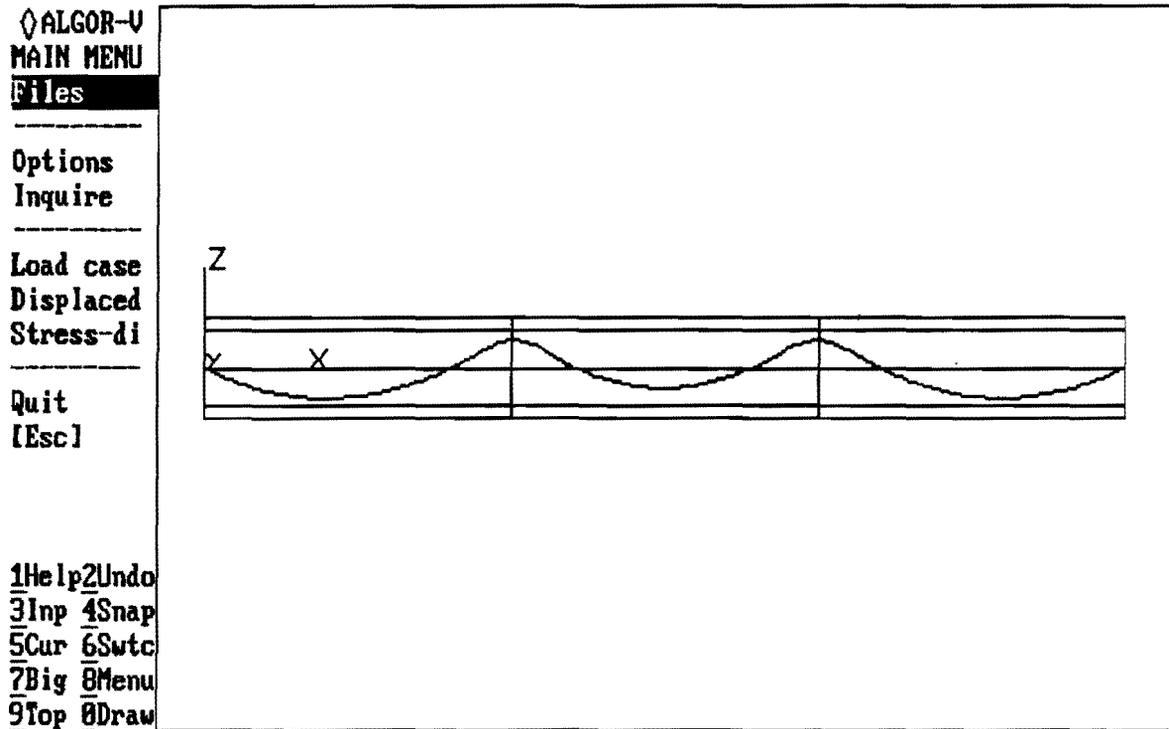


FIG. 14. Profile View of Sample Problem

When finished viewing the model in the profile view, select **Quit** from the MAIN MENU of SuperView. Return to the *TEXSLAB* program by selecting the *TEXSLAB* icon at the bottom of the screen. Press [Esc] to remove the box from the screen.

7.4.2 Postview

If the analysis has not been performed, as mentioned earlier, graphical results can still be viewed but the **Postview** option is not available. Select **Preview** from the VIEW menu. Since SuperView runs independently from *TEXSLAB*, if the data files exist they can be viewed. Follow the procedure below to view results once SuperView has been activated.

After the analysis has taken place (see section 7.5), results can be viewed graphically. In a manner analogous to the steps used to select **Preview**, select **Postview**. Assuming the analysis has been completed, another menu box appears

with three choices: **Top Stress**, **Bottom Stress**, and **Middle Stress**. All three options are available all the time. Select **Top Stress** and a box appears on the screen instructing the user to press **[Ctrl] + [Esc]**; do so. As in **Preview**, the selection must be made so that the proper files are arranged for viewing in SuperView. Select **ALGOR-SuperView** from the *TEXSLAB* group. As in **Plan View** under **Preview**, the same model appears on the screen. SuperView is again instructed by the file **SAMPLE.SVD** on how to display the model and what to display in the postprocessing mode. For the sample problem, SuperView displays stresses in the global X direction. Refer to Appendix II for more details on postprocessing the model. To view the resulting stresses, do the following:

1. Select **Stress-di** from the MAIN MENU.
 - Note that an asterisk is by option **Post**. This means that results of stresses and displacements are shown by means of dithered contours on the model.
 - Note that an asterisk is by option **Smooth**. This means that results are smoothed over each element rather than having distinct contours.
 - Note a rectangular (white) box appears in the upper right portion of the screen. This is the legend box where values are assigned to colors.
2. Select **Do dither** from the STRESS-DI menu.
 - The stresses are read in and color patterns cover the model representing the stress magnitudes.
 - Note the two lines under the display window (Fig. 15). The first line indicates what is displayed. For the sample session this is:

Dither method = Tensor Stress Dot vector: X=1, Y=0, Z=0.

Thus, the stress is given in the direction of the vector (1,0,0). This is the stress σ_x as seen in Fig. 12.
 - The second line shows the extremes for the model.

The displacements can always be viewed regardless of what is selected from the **Postview** submenu. To view displacements from the NOPARC analysis, do the following:

1. From the MAIN MENU within SuperView, select **Stress-di**.
2. Select **Post**.
3. Select **disp Vec**.

4. Select **Dot**.
5. Select **Z dir**. This instructs SuperView to display vertical displacements.
6. Press [Esc] three times and select **Do dither** from the STRESS-DI menu.
 - The displacements are read in, and colors representing the stresses cover the model.
 - Note the two lines under the display window (Fig. 16). The first line indicates what is displayed. For the sample session this line is:
Dither method = Vector: Translation Dot vector: X=0, Y=0, Z=1.
 Thus, the displacement is given in the direction of the vector (0,0,1).
 - The second line shows the extremes for the model.

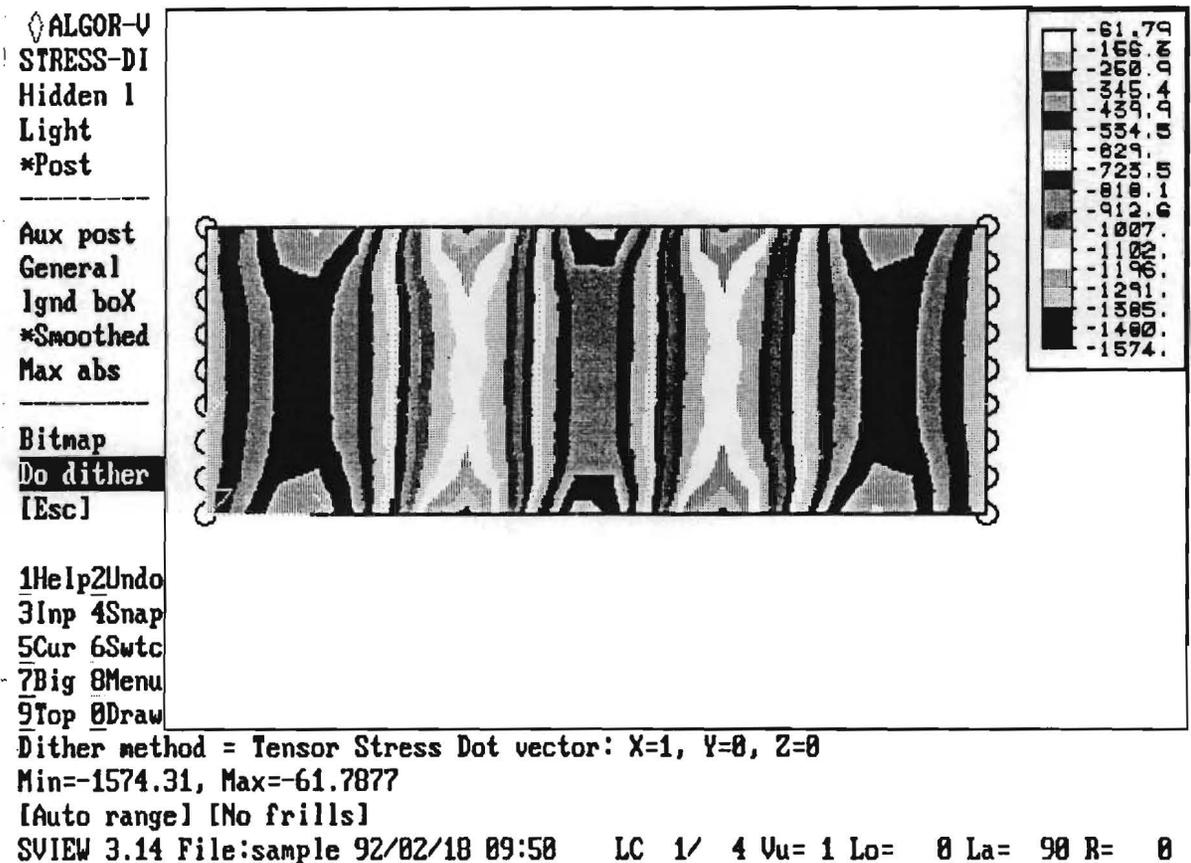


FIG. 15. Stress Dither of Sample Problem

7.5 Analyze

After checking the plan and profile views from within the **Preview** submenu, it is appropriate to analyze the model. As mentioned before, two graphical output files accompany the program. This set may be skipped to conserve time for the

analysis. The included files contain the displacement values for the problem and stress values for the top layer of the concrete. Refer to section 7.4.2 and apply the procedure used for **Postview** if **ANALYZE** is to be skipped. If not, select **ANALYZE**. This opens a submenu from which **Run** is selected. When **Run** is selected, the analysis program **NOPARC** is started. The sample problem requires 20 MB of hard disk space and runs for 30 minutes on an IBM Model 80, 25 MHz machine. When the status box in the lower right corner indicates that **NOPARC** is "Done," return to section 7.4.2 and go through the postview procedure.

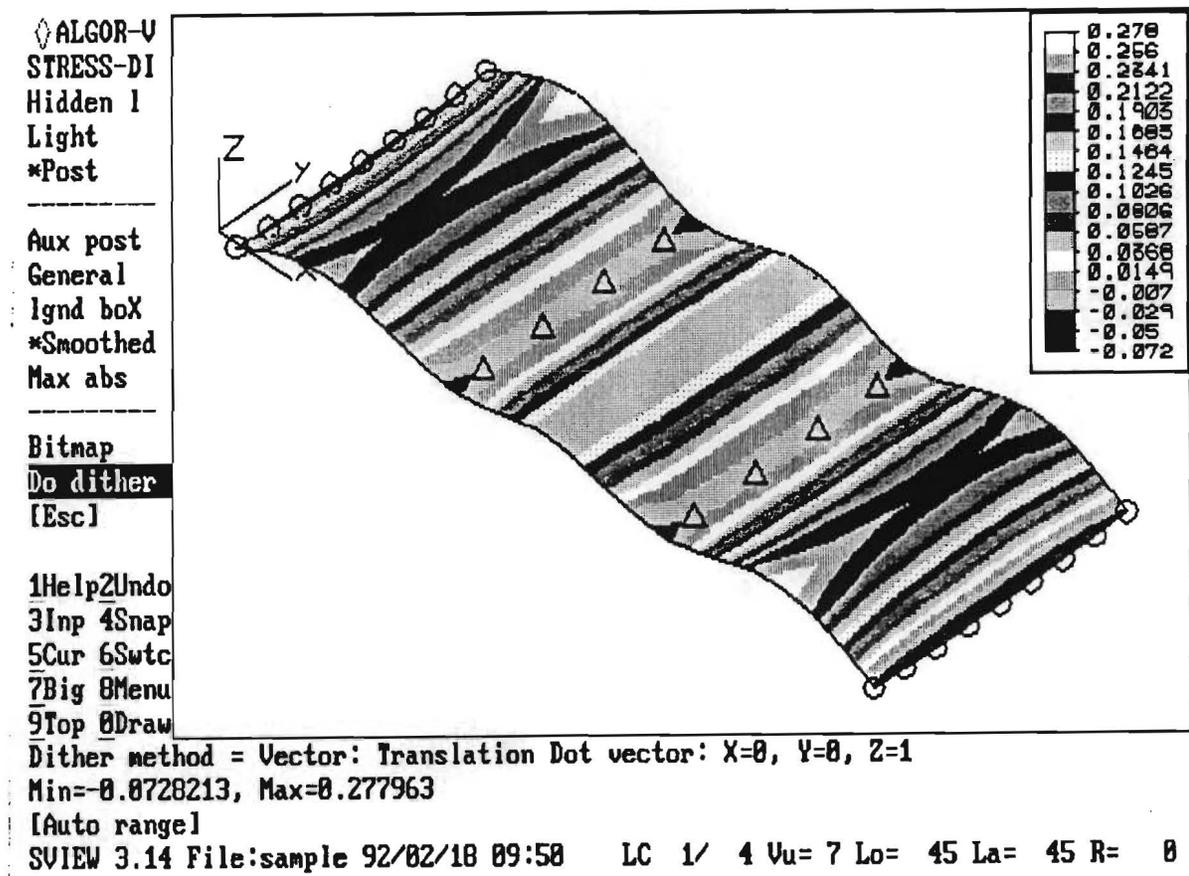


FIG. 16. Displacement Dither of Sample Problem

8. WORK FILES

To operate, *TEXSLAB* requires two data files in addition to the files which comprise the *TEXSLAB* program package that are listed in section 3.2. The primary files required in preparation for execution of the analysis program are called microfiles. Default microfiles are provided with the program. Many additional files are created by *TEXSLAB*. All files for a problem have the same prefix name, with the exception of the file created to represent the profile of the slab. This file is identified as **PROFILE**. The remaining files have names that vary only by the extension. All files are discussed below.

8.1 Default Microfiles

- **DEFAULT.IFL:** This file is required for initial operation of *TEXSLAB* when no previous input microfile exists. Default values are read from this file and are used in the input fields for bridge input when a new problem is started. The default input microfile cannot be transformed into a NOPARC input file because no geometrical dimensions are specified. This file may be modified by opening it using the **Open** command in *TEXSLAB*. Type **DEFAULT** when prompted for the file name to open. Edit the file using the **Edit** menu and **Bridge Input** submenu. Escape out of the **Edit** menu and the old default file is overwritten by the newly edited file.
- **DEFAULT.OFL:** This file is required for initial operation of *TEXSLAB* when no previous output microfiles exist. Default values are read from this file and used in the input fields to specify bridge output when a new problem is started. This file may be modified as discussed for the input microfile. Select **Bridge Output** instead of **Bridge Input**.

8.2 Input Microfiles

Input microfiles refer to those that have been previously created or that are about to be created. Previously created files are those that have already been worked on in some fashion within *TEXSLAB*. Once a new file name has been specified in the **New** submenu, an input and an output microfile exist. Default microfiles are copied to new files with names specified in the menu box. If the program is stopped, these files will exist. The name specified in any of the **File** submenu boxes is referred to as *filename* in the discussion below.

- ***filename.IFL***: This is the microfile for input parameters. A default microfile exists that contains many default values for input file generation. The editor within *TEXSLAB* reads and writes to the microfile. All data displayed on the screen is contained in the microfile. Information in the microfile is taken and expanded into an input file for NOPARC by use of the TRANSFORM menu. This file is in alphanumeric format and can be viewed with any text editor. Modifications can be made to this file outside of *TEXSLAB*.
- ***filename.OFL***: This file contains information that controls the output of NOPARC. This file is very similar to the *.IFL* file.

8.3 Transform Output Files

When TRANSFORM is run, three alphanumeric files are created. One is the input data file for the analysis. The other two are for purposes of graphical representation.

- ***filename.WFL***: This is a temporary file used by TRANSFORM. It is deleted if no errors are encountered when the microfiles are read. If an error is found in the input, check this file to determine the problem.
- ***filename.DAT***: The *.DAT* file is the input file for NOPARC analysis. The format of this file is specific and is given in the reference manual (van Greunen 1979). It is a large alphanumeric file containing information that is required to perform the analysis. A text editor can be used to modify this file outside of *TEXSLAB*.
- ***filename***: This file does not have an extension. This file is created for purposes of graphical representation. It is converted to a binary file when **Plan View** is selected from the VIEW menu followed by selecting **ALGOR-SuperView** from the *TEXSLAB* group in the OS/2 Presentation Manager screen. The newly created binary file created for SuperView is named *filename.SST*. The plan view of the model comes from this file.
- **PROFILE**: This file does not have an extension. This file is created for purposes of graphical representation. It is converted to a binary file when **Profile View** is selected from the VIEW menu followed by selecting **ALGOR-SuperView** from the *TEXSLAB* group in the OS/2 Presentation Manager screen. The new binary file created for SuperView is named *PROFILE.SST*. The profile view of the model comes from this file. This file is overwritten whenever TRANSFORM is run. Care should be taken to save this file if desired.

8.4 Analysis Output Files

The analysis program NOPARC creates five output files. One contains alphanumeric results, and the other four are binary files used for graphical representation. The size of these files can be very large and is dependent on the problem size. Size can be kept minimized by making judicious selections in the **Bridge Output** submenu.

- **filename.OUT:** This file contains the alphanumeric output that is specified in the .OFL file. Partial output of the input file is included at the beginning of this file. Detailed tendon profile information follows an echo of the input data. The general output includes information pertaining to the convergence of the analysis, values given at the nodes, and values within finite elements. Specification of the frequency of output and the content of this output is done in **Bridge Output**. The following information is output in a group for each output step. The output step specified in **Bridge Output** may include every iteration, every load step, or only the output at the end of all load steps for a particular analysis. Nodal forces and displacements are output next for those nodes specified. Element stresses and strains are output following the nodal information for those elements specified. Support reactions are output at the end of the output for the last element. The output contained in this file may be viewed with a text editor or printed. Depending on what is specified, this file can be very large.

- **filename.DO:** This is a binary file created for graphical viewing of the displacements in SuperView. It does not require any conversion or name changing for SuperView to use it. Start SuperView as discussed in section 4.4 or type

SVIEW filename

at the DOS prompt. This file is read when the deflected shape is shown or displacements are shown in a dithered plot.

- **filename.ST:** This is a binary file created for graphical viewing of stresses occurring in the top layer of the concrete. This file must be copied to

filename.NSO

before SuperView can use it. Do not use the DOS RENAME command to change the name. The middle and bottom layer stress

files must also have the same name when SuperView reads them. Thus, the original file may be deleted if it is renamed and then overwritten with a different layer's file. *TEXSLAB* renames this file when **Top Stress** is selected in the **Post View** menu. This file is read when stresses are shown in a dithered plot.

- ***filename.SM***: This is a binary file created for graphical viewing of stresses occurring in the middle layer of the concrete. This file must be renamed to

filename.NSO

before SuperView can use it. The top and bottom layer stress files must also have the same name when SuperView reads them. Thus, the original file may be deleted if it is renamed and then copied with a file from a different layer. *TEXSLAB* renames this file when **Middle Stress** is selected in the **Post View** menu. This file is read when stresses are shown in a dithered plot.

- ***filename.SB***: This is a binary file created for graphical viewing of stresses occurring in the bottom layer of the concrete. This file must be renamed to

filename.NSO

before SuperView can use it. The top and middle layer stress files must also have the same name when SuperView reads them. Thus, the original file may be deleted if it is renamed and then copied with a different layer file. *TEXSLAB* renames this file when **Bottom Stress** is selected in the **Post View** menu. This file is read when stresses are shown in a dithered plot.

APPENDIX I. OPERATING TEXSLAB IN OS/2

Depending on which version of OS/2 is installed, one of two procedures can be used to set up *TEXSLAB* for execution on a microcomputer. Each procedure is described in detail in the following sections.

OS/2 Version 1.1

Setup

No prior setup configuration is required if OS/2 version 1.1 is used.

Execution

TEXSLAB will operate in OS/2, version 1.1, but will not be as user-oriented as it is with newer versions of OS/2. The procedure to start *TEXSLAB* is outlined as follows:

1. With OS/2 version 1.1 up and running, the Main Group in the Start Programs window should be on the screen.
2. Select **OS/2 full screen command prompt** from the Main Group. This brings up a full screen command prompt similar to that of DOS.
3. At the prompt, type the following command to go to the directory where *TEXSLAB* is installed:

```
[C:\] CD TEXSLAB.
```

4. To begin *TEXSLAB* enter:

```
[C:\TEXSLAB] TEXSLAB.
```

5. When preview or postview of graphical information is selected, the program prompts the user to press [Ctrl] + [Esc] and select **DOS**.
6. The DOS shell is now loaded. Get into the *TEXSLAB* directory.

```
C:\> CD TEXSLAB.
```

7. To view the model:

C:\> TEXSLAB\ STARTSV.

8. SuperView should be running now. When finished examining the graphical view of the model, select **Quit**. This returns the user to the DOS command line.
9. Press **[Ctrl]+[Esc]** and select the OS/2 icon at the bottom of the screen to return to *TEXSLAB* menu. Selecting **OS/2 full screen command prompt** from the Task Manager also works. Do not select **OS/2 full screen command prompt** from the Main group. Doing this would start an additional OS/2 full screen command prompt.

OS/2 Versions After Version 1.1

The procedure outlined above also is compatible with newer versions of OS/2, but the following discussion provides a user-oriented approach. The OS/2 program can be set up to run *TEXSLAB*. When OS/2 is running, the Desktop Manager window appears on the screen. Desktop Manager is used to configure OS/2 to operate *TEXSLAB*. The procedure to do this is outlined next in Setup:

Setup

Create a *TEXSLAB* Group.

1. From OS/2 Desktop Manager, select **G**roup from menu bar.
2. Select **N**ew.
3. Type *TEXSLAB* for new group name.
4. Select **A**dd.

Create a *TEXSLAB* program within the Group *TEXSLAB*.

1. Select **P**rogram.
2. Select **N**ew.
3. Type *TEXSLAB* for program title.
4. Press **[Tab]** to get into the box for path and filename, and type the following, assuming *TEXSLAB* is installed on drive C:

C:\TEXSLAB\TEXSLAB.EXE.

5. Press **[Tab]** twice to get to the working directory box and type,

C:\TEXSLAB.

6. Select **A**dd.

Create program for ALGOR SuperView within the Group *TEXSLAB*.

1. Select **P**rogram.
2. Select **N**ew.
3. Type ALGOR SuperView for program title.
4. Press [Tab] to get into box for path and filename, and type the following, assuming SuperView is installed on drive C.

C:\TEXSLAB\STARTSV.EXE.

5. Press [Tab] twice to get to the working directory box and type,

C:\TEXSLAB.

6. Select **A**dd.

The above procedures only need to be performed when *TEXSLAB* is first installed on the computer. To operate after installation, refer to the following procedure.

Execution

1. From OS/2 Desktop Manager, select the *TEXSLAB* icon as the program group.
2. Select *TEXSLAB* by clicking with the mouse on the OS/2 icon next to *TEXSLAB*. Doing this starts the program.
3. When **Preview** or **Postview** from the **View** menu is selected, the program prompts user to press [Ctrl] + [Esc] and select **ALGOR SuperView**.
4. The Task Manager appears. Select **Group - TEXSLAB**.
5. Select **ALGOR SuperView** to get into the DOS mode and start the SuperView program.
6. When finished with SuperView, select **Quit**.
7. Press [Ctrl] + [Esc] and select *TEXSLAB* from Task List or select the icon at the bottom of the screen for *TEXSLAB*.

APPENDIX II. BACKGROUND AND COMMANDS FOR SUPERVIEW

SuperView is a commercially available graphical software package that is used in conjunction with NOPARC to display model geometry and analysis results. A comprehensive user's manual is available for the program and has been used in preparing this report. A complete listing of manuals related to this software is included in the References (*ALGOR ViziCad* 1990; *ALGOR ViziCad plus* 1990). The following discussion highlights commands that are useful and, in most cases, sufficient for pre- and postviewing the model.

General SuperView Control

Convention

To aid in using this outline of commands, some conventions need to be explained. All relevant keys on the keyboard with specific names will be enclosed within angle brackets. Some examples are <F3>, <Esc>, and <Enter>. When referencing a menu within SuperView, the menu will be written in all capital letters; MAIN MENU is an example. The word 'select' is used in two ways. When select is followed by a statement in **bold text**, the user is asked to select a menu option with the mouse or keypad. When select is used but not as in the previous case, the user is asked to select something on the model or select from a list of options within the menu. This is also done with mouse or keypad. The word 'enter,' when not enclosed in angle brackets, prompts the user to enter something from the keyboard. When 'activate' is used, the user is requested to select the **bold text** item following it. Activating the statement places an asterisk in front of it. Deactivating a statement can be done by selecting an already activated item.

Input Device

SuperView is a menu-driven graphical program. A mouse is strongly recommended but not required for program operation. Specification of the type of input device is required for proper menu selection. This specification should correspond to the way SuperView was configured during installation in section 3.2. With SuperView started, the user can specify as follows (see *ALGOR ViziCad*, vol. 1 1990, sec. 3.3):

1. Press <F3> function key on keyboard or use mouse to click on **3Inp** which is located in the lower left hand portion of screen. This action brings up the INPUT menu.
2. If a mouse is available, activate the **Mouse/Tab** option. If a mouse is not available and the user wishes to move around the screen with arrow keys, activate the **E kEypad** option. If a mouse is not available and the user wishes to input coordinates of points using the keyboard, activate **Keyboard**.
3. When **Keyboard** is selected, the user must specify if the keyboard input is **Relative** or not and what coordinate system is being used.
4. Press the <Esc> key or click on **Esc** when finished.

To select menu options with the mouse, highlight the selection by moving the white cursor bar to the desired location; press the left button on the mouse. For **E kEypad** input, select a choice by moving the white cursor bar over the choice with the arrow keys and press <Enter>. SuperView also incorporates extended keypad commands for the **E kEypad** mode. The user is referred to *ALGOR ViziCad, vol. 1* (1990) for information on the extended keypad. Regardless of which input device is selected, menu selections can be made by pressing the capital letter that appears in each menu option that is not supported with a function key. The SuperView screen is arranged so that all menu choices appear on the left side. The menu options that can be selected by pressing a keyboard letter comprise the upper portion of the screen and those selected by a function key, the lower portion of the screen.

Help Information

Help is available within SuperView for all commands and menu options. When help is wanted, press <F1> and select the command or menu option. Pressing <Enter> or <Esc> after reading the help information returns the user to the menu of origination (*ALGOR ViziCad, vol. 1* 1990, sec. 3.4.3).

Zoom Procedure

A bridge model typically is large and contains many components. Examining specific aspects of the model may be difficult because they can be very small compared to the total model. Portions of the model can be enlarged so that more detailed inspection can take place using the zoom feature. To zoom in on a portion of the model, follow this procedure (*ALGOR ViziCad, vol. 1* 1990, sec. 3.5):

1. Press <F10> to activate the DRAW menu, or select **0Draw**.
2. Select **zoom In**.
3. Box in the area that is to be zoomed. After selection is made, the selected area occupies the entire screen.
4. To zoom in even more, repeat steps 1 through 3.
5. For the entire model to be redisplayed, select **Enclose** from DRAW menu.

Preview Procedures

View Options

After **Preview** has been activated from the presentation manager, SuperView displays the model on the screen. The manner in which the model is displayed may not be suitable to the user. If the model appears on the screen rotated through some angle and the user wishes to view the bridge differently, the following procedure can be followed.

1. Select the DRAW menu. Either press function key <F10>, or select **0Draw** which is located in the lower left portion of screen.
2. Select **View**.
3. Select which view is desired from the list. Option 1 (XY Top) works well for the plan view and XZ Front for the profile view.
4. After the requested view is chosen, the model is redrawn. It may no longer fit within screen boundaries. Select **Enclose** from the DRAW menu to get the model to fit within the screen.
5. Press <Esc> to get back to the MAIN MENU if satisfied with the current view of the model.

Constraint Display

All nodes within the model are restrained against rotation about the z-axis. Therefore, all nodes have circles around them showing that there is some constraint present (*ALGOR ViziCad plus, vol. 2 1990, sec. 7.3.3*). If the user wishes to see only the nodes constrained uniquely, such as those at abutments and columns, perform the following steps.

1. From MAIN MENU, select **Options**.

2. Select **Node +pres**.
3. Select **bc masK**.
4. Select boundary conditions to display. Six constraints are listed: translation in x-direction, translation in y-direction, translation in z-direction, rotation about x-axis, rotation about y-axis, and rotation about z-axis. As mentioned before, rotation about the z-axis is constrained at all nodes. For column supports, in-plane constraints are placed in both the x- and y-directions in addition to vertical support in the z-direction. Abutments supply constraint in the z-direction. Activate or deactivate constraints to be displayed or ignored.
5. Press <Esc> three times to return to MAIN MENU.
6. Press <F10> or select **0Draw**.
7. Select **Redraw** to show the model with updates for changes made.

Model Colors

If the model contains prestressing tendons, the user has an option to make the color of the tendons differ from that of the bridge. Prestressing tendons are graphically represented by truss elements, and the concrete slab is represented by plate elements. To change the color of either type of element:

1. From MAIN MENU, select **Options**.
2. Select **Ele opt**.
3. Select **1) truss** or **6) plate**.
4. Select **Color**.
5. Using the keyboard, enter the number corresponding to the color desired. Numbers for colors can be seen by looking at the function-key menu selections. Ten options are available with a different color bar under each option number (1 to 10), where 10 is represented by 0. Press <Enter> when the desired color has been selected.
6. To see the color change, press <F10> or select **0Draw**.
7. Select **Redraw**.
8. Press <Esc> once to get back to the ELE OPT menu if additional color changes need to be made. Repeat steps 3 through 7.
9. Press <Esc> three times to return to the MAIN MENU.

Model Inquiry

SuperView can be used to check whether input from the user agrees with the generated output for the bridge. Section 4.4.1 lists several things to check for in the plan view and the profile view. They are discussed below:

Plan View

A. Basic geometry of the bridge and discretization of the model primarily refer to the location of the four corners of the bridge, number of mesh divisions longitudinally and transversely, and element thickness.

To check the location of the bridge corners, do the following.

1. Select **Inquire** from MAIN MENU.
2. Select **Node BC**.
3. Activate **Get**.
4. Move the pointer to a corner of the bridge and push the left button on the mouse, or with the keypad press <Enter>. Note X and Y coordinates are displayed at the bottom of the screen.
5. Repeat for the remaining three corners of the bridge slab.
6. Press <Esc> two times to return to MAIN MENU.

Check the number of divisions longitudinally and transversely by visually observing the model and counting the divisions. To check the total number of elements and nodes, do the following.

1. Select **Inquire** from MAIN MENU.
2. Select **stats**.

Note: Details of the model are displayed beneath the window screen. The number of nodes, number of load cases, and type and number of elements are listed. Check whether they are the same as the inputs given with *TEXSLAB*. Element group 1 (G1), element type 6 (ty=6) (plate elements), describes the number of elements that discretize the concrete slab.

To check the element thicknesses in the plan view, do the following. It can be noted that slab thickness may be better viewed in the profile view.

1. Select **Inquire** from MAIN MENU.
2. Select **Ele inq.**
3. Select **6) plate.**
4. Activate **Get.**
5. Move the pointer to the center of an element and select with the mouse or arrow keys; then press the left mouse button or press <Enter>, correspondingly. If the mesh is very dense, it may be difficult to select a particular element. The user may wish to use the zoom procedure discussed earlier to aid in selection of the element in question.
6. Check the thickness by looking at the top line in the bottom portion of the screen. Thickness is given as Thick = *thickness*.
7. When finished checking the element, press <Esc> three times.

Note: If a distributed load is applied to the bridge, the pressure on the selected element can be observed in a manner that is similar to the thickness check in step 6. Pressure is given by Press = *pressure*.

B. To verify that supports for the slab are in the correct location, check node locations where either a circle or a triangle surrounds the node. Only nodes that are uniquely constrained are enclosed if the previous procedure for displaying constraints has been followed. To verify support locations perform the following:

1. From MAIN MENU, select **Inquire**.
2. Select **Node BC.**
3. Activate **Get.**
4. Select the node in question as was done for geometric inquiry of the corner nodes. Note that coordinates of node and constraint conditions are listed at the bottom of screen.
5. Select other nodes with nodal constraint.
6. Press <Esc> two times to return to MAIN MENU.

C. Checking of the tendon configuration is important to assure that the model is defined as expected. The total number of tendons, tendon spacing, and tendon location can be determined by the following steps, beginning with the number of tendons.

1. From MAIN MENU, select **Inquire**.

2. Select **Stats**. Information pertaining to the model is given at the bottom of the screen. The number of tendons is given in the second line. Element group 2 (G2), element type 1 (ty=1) (truss elements), describes the number of tendons.

To check tendon spacing, it is advisable to zoom in on a few of the tendons where they intersect the bridge boundary. After zooming, perform the following steps.

1. Select **Inquire**, from MAIN MENU.
2. Activate **Distance**.
3. Click with the right button of the mouse or select with the keypad at the end of a tendon.
4. Move to the next tendon in the group and repeat step 3. The distance between two tendons defines tendon spacing and is given at the bottom of the screen. DS denotes the total distance. Component distances DX and DY are also given.
5. Repeat steps 3 and 4 for other tendon groups. The model may need to be redrawn to full scale, and new portions can be zoomed in on.
6. Press <Esc> once to return to MAIN MENU.

Checking tendon location with respect to the rest of the bridge is important to insure that tendon groups are in the correct location. For example, initial and final distances of the tendon from the edge of the bridge are important and should be checked.

1. Repeat steps 1 through 4 in the tendon spacing procedure.
2. For the second point, select the node at the desired distance from the tendon.
3. Repeat steps 1 and 2 if other tendons need to be checked.

Checking the applied tendon force is discussed below. The arrows extending inward from the boundaries of the model represent the applied tendon force. The applied force is the same for all tendons within a group; thus, checking a single tendon of each group is sufficient.

1. Select **Inquire** from MAIN MENU.
2. Select **Force**.
3. Activate **Get**. Note that there should be an asterisk by **Get** when activated.

4. Select a node on the model with a force extending from it. The node number selected, the force components, and the magnitude of the force are given at the bottom of the screen.

Note: Regardless of how the applied tendon force appears on the screen, the force is always applied in the direction of the tendon. For longitudinal tendons, the force component in the global X direction (F_X) is used for representation, and for transverse tendons, the force component in the global Y direction (F_Y) is used.

D. The load configuration consists of two parameters. The first is the location and magnitude of the loads applied. The second is when the loads are applied. Each is discussed below.

Concentrated and distributed loads are displayed by SuperView if the correct view is chosen. When the XY Top view is selected, out of plane loads appear as a point in the mesh. Distributed loads are modeled as pressure on each element face. (Determination of the value of the pressure is discussed in finding the bridge thickness, and the user is referred to that section.) Concentrated loads are at nodes and are not visible. A different view is needed to display them. Follow these steps to determine concentrated load information.

1. Repeat steps 1 through 5 as explained in View Options. This time, however, pick view option 7 (**Isome**).
2. **Zoom In** on the area where a concentrated load is applied.
3. Select **Inquire** from MAIN MENU.
4. Select **Force**.
5. Activate **Get**.
6. Select the node from which the force originates. The node number, components of force, and force magnitude are displayed at the bottom of the screen.
7. Repeat steps 2 through 6 for other concentrated loads.
8. Press <Esc> twice to return to MAIN MENU.

Since many analyses can be specified by *TEXSLAB*, the application of load can occur at different times. In SuperView, the number of analyses is referred to as the number of load cases. The number of load cases (analyses) is shown at the bottom to the screen as *LC (load case number)/(number of load cases)*. When SuperView is started, the first load case is displayed. If there are prestressing

tendons, the applied prestressing force always shows up on the first load case. To view the loads applied for other load cases, the steps are as follows:

1. Select **Load case** from the **MAIN MENU**.
2. Select **Next** or any of the other applicable options to select a new load case.
3. Push <F10> to get the **DRAW** menu.
4. Select **Redraw** to redraw the model and display any newly applied loads.
5. Repeat for all load cases.

Note: It should be made clear that once a load is applied, it is always applied. Once the load appears on the display for a single load case, it is not shown for all remaining load cases even though the load is still applied. Thus, only the newly applied loads are shown for each loadcase. For time-dependent analysis, no loads are shown as being applied to the model.

Profile View

SuperView behaves identically in the profile view as it did in the plan view. A important thing to note is that the view displayed on the screen for profile view should be the XZ front view. See View Options under preview options above for instruction on how to obtain the XZ front view. Also, it should be noted that all Z-components in the profile view are scaled up by a factor of ten. Several SuperView procedures that are useful for the profile view are listed as follows:

A. The relative location of individual regions can be checked. Regions within a model are separated by vertical lines extending from the top of the slab to the bottom. A vertical line always separates two spans. This line is not different from the division lines between regions. To find the location and size of each region, follow the steps given below:

1. Select **Inquire**, from **MAIN MENU**.
2. Activate **Distance**.
3. Click with the right button of the mouse or select with the keypad at the intersection of a vertical line with either the top or bottom boundary line. The first line under the display window shows the global coordinate of the point selected. Remember that this is a cross-section taken at a tendon line; therefore, if the structure is skewed, the numbers may not be as expected.

4. Move to the beginning of the region and repeat step 3. The length of the region is shown on the second line beneath the display window. DS denotes the total distance. Component distances DX and DY are also given.
5. Repeat steps 3 and 4 for other regions.
6. Press <Esc> once to return to MAIN MENU.

B. Slab thickness, reinforcing steel location, and the prestress tendon eccentricities may be checked while in the profile view. Verification of all these quantities may be done using the following procedure.

1. Select **Inquire** from MAIN MENU.
2. Activate **Distance**.
3. Click with the right button of the mouse or select with the keypad at the intersection of the vertical line with the centerline on either the right or left boundary. The global Z coordinate at the centerline should be zero ($Z=0$.); verify this. This can be viewed on the first line under the display window. With the centerline as $Z=0$., the elevation of all other points of interest can be found with respect to it. The center line is referred to as the reference plane.
4. Move the mouse to the point of interest. If the tendon is being checked, click the right button of the mouse to find the Z coordinate at that location. If the top or bottom of the slab or a steel layer is being checked, click the right button of the mouse near the intersection of a vertical line and the concrete boundary or steel layer, respectively. Clicking the right button instructs SuperView to select the nearest node. If a location is desired other than the above, select it using the left button of the mouse. The Zoom Procedure outlined previously may be useful. The elevation (Z coordinate) with respect to the reference plane is given on the first line under the display window.
5. Repeat step 4 for other points of interest.
6. Press <Esc> once to return to MAIN MENU.

Postview Commands

After the analysis is complete, the results can be presented graphically with SuperView. Bridge deflections, concrete stresses, and tendon forces can be seen graphically. Procedures for displaying the previously mentioned parameters are discussed in what follows.

Bridge Deflections

Deflections can be observed graphically in two ways. First, the procedure for observing the deflected shape of the bridge is discussed. Second, the method of displaying nodal displacements with color plots is discussed. The following steps cause the deflected shape to be displayed (*ALGOR ViziCad plus, vol. 2 1990, sec. 10.3.1*).

1. From MAIN MENU, select **Displaced**.
2. Activate **Displaced** to see the deformed shape.
3. A different view of the model may aid in viewing the deflected shape. If desired, select a different view from the DRAW menu (<F10>) as explained previously.
4. If the undeformed model is also to be shown, activate **With und**.
5. If the deformed shape requires more exaggeration, select **Scale** if scale is to be selected by user, or **Calc scal** if the scale is to be calculated automatically.
6. Individual nodes are subject to inquiry from **Displaced** menu. Select **Nodes inq**.
7. Select a node at which deflections are desired. Coordinate displacements are listed at the bottom of the screen.
8. Press <Esc> twice to get back to MAIN MENU.

To show displacements with color plots, follow the procedure outlined next (*ALGOR ViziCad plus, vol. 2 1990, sec. 10.10*).

1. From MAIN MENU, select **Stress-di**.
2. Select **Post**.
3. Select **disp Vec**.
4. Activate **Magnitude** if the displacement plot is based on the magnitude of the displacement vector. Activate **Dot** if the displacement plot is based on the dot product of the displacement vector with the entered vector.
5. If **Dot** is activated, select **Vector**.
6. Select vector or create a custom vector that is to be dotted into the displacement vector. The **Z-dir** vector gives the vertical displacement.

7. Press <Esc> four times and select **Do dither** to display the displacement plot.
8. Configure plot to fit preferences as described at the end of the explanation of the postview commands.

Concrete Stresses

SuperView provides many ways to view stress results. The user is referred to *ALGOR ViziCad plus, vol. 2 (1990)* for all stress display methods that are available. Stress components in the global X and Y directions are discussed in what follows:

1. Select **Stress-di** from MAIN MENU.
2. Select **Post**.
3. Activate **S tensor**.
4. Select **Vector**.
5. Select either **X dir** or **Y dir** to view stresses in the global X or Y directions, respectively.
6. Press <Esc> three times.
7. Select **Do dither**. Color plot is displayed.
8. Configure plot to fit desired preferences as explained at the end of the postview commands.

Tendon Force and Stress

Forces and stresses occurring in tendons can be viewed by the user as follows (*ALGOR ViziCad plus, vol. 2 1990, sec. 10.10*).

1. From MAIN MENU, select **Stress-di**.
2. Select **Post**.
3. Activate **Beam-trus**.
4. Activate **P/A** for tendon stress or **r1 axf** for tendon force.
5. Press <Esc> twice.
6. Select **Do dither**.
7. Configure plot to fit preferences as explained at the end of postview commands.

Postview Plot Configuration

A suitable method for displaying color plots on the screen is outlined here. Other configurations are available and different users may prefer them, but for the sake of example, the method and configuration that has been successfully used is

described. A detailed explanation of each command can be found in *ALGOR ViziCad plus, vol. 2* (1990).

Display Range Values

To insure that all analysis values are displayed completely on the screen, the following procedure can be followed (*ALGOR ViziCad plus, vol. 2* 1990, sec. 10.11).

1. From STRESS-DI menu, select **Aux post**.
2. Activate **Auto rng** if not sure about the range of analysis results.
3. Redo **Do dither** to display plot with changes made.

Speed Up Dithering

To speed up dithering of plots on the screen, do the following (*ALGOR ViziCad plus, vol. 2* 1990, sec. 10.5.1).

1. From STRESS-DI menu, select **General**.
2. Activate **Solid-di**. Colors will be distinct for each range; change of color from one range to the next is abrupt.
3. Press <Esc> once.
4. Select **Do dither**.

Assign Colors

Allowing the maximum number of colors to be displayed gives the best resolution for results. To assure that the maximum number of colors are used, do the following (*ALGOR ViziCad plus, vol. 2* 1990, sec. 10.11).

1. From STRESS-DI menu, select **Aux post**.
2. Select **Colors**.
3. Select **Number**.
4. Assign number to 16.
5. Select **Sequence** to automatically assign colors to ranges.

Modify Legend Box

The number of colors in the legend box should correspond to the number of colors that are assigned. Additional modifications to the legend box can also be as outlined in the following (*ALGOR ViziCad plus, vol. 2* 1990, sec. 10.8.1):

1. Select **lgnd boX**.
2. Select **Num lines**.

3. Enter the number of value lines assigned in the legend box. This number should be equal to the number of colors set for the dither display plus one.
4. Change the number of digits displayed and size of the text with **Fmt chars** and **Text size**, respectively.
5. Specify the position of the legend box on the screen by choosing a location specified in this menu.

APPENDIX III. EXAMPLE MICROFILES

Input Microfile - SAMPLE.IFL

```

*-----
* Geometric Parameters
*-----
CORNER POINTS OF THE SLAB
*
* (in counterclockwise order,
*  starting from left-bottom corner)
*
* point | x | y |
* number | coordinate | coordinate |
* 1 | .00 | .00 |
* 2 | 150.00 | .00 |
* 3 | 150.00 | 56.00 |
* 4 | .00 | 56.00 |
*
COLUMN CONTROL PARAMETERS
* Number of column lines:
  2
* Number of columns in a line:
  4
* Number of mesh divisions transversly:
  8
*
COLUMN LOCATIONS
*
* column line #1
*
* column | x | y |
* number | coordinate | coordinate |
* 1 | 50.00 | 7.00 |
* 2 | 50.00 | 21.00 |
* 3 | 50.00 | 35.00 |
* 4 | 50.00 | 49.00 |
*
* column line #2
*
* column | x | y |
* number | coordinate | coordinate |
* 1 | 100.00 | 7.00 |
* 2 | 100.00 | 21.00 |
* 3 | 100.00 | 35.00 |
* 4 | 100.00 | 49.00 |
*
GENERAL SPAN INFORMATION
* Span # | Number of regions in span
  1 | 1
  2 | 1
  3 | 1
*
SPAN SPECIFIC INFORMATION
* Region | Location | Number of
* Number | of beginn | division in region |
* Span Number | 1 | 7
* Span Number | 2 | 7
* Span Number | 3 | 7
* Span Number | 1 | 7
*
SPRING SUPPORT PARAMETERS
*Verticle spring stiffness at abutments ((k/ft)/ft):
  1000000.00

```

*Longitudinal spring stiffness at abutments ((k/ft)/ft):
 .00
 *Transverse spring stiffness at abutments ((k/ft)/ft):
 28800.00
 *Verticle spring stiffness at columns ((k/ft)):
 8520000.00
 *Longitudinal spring stiffness at columns ((k/ft)):
 42500.00
 *Transverse spring stiffness at columns ((k/ft)):
 72500.00

*
 *-----
 * Concrete
 *-----

CONCRETE DATA

* compressive strength at 28 days
 6300.00
 * Tensile strength
 -1.00
 * Poissons ratio
 .15
 * Specific weight
 145.00
 * slump in inches
 7.10
 * Number of concrete layers in slab?
 1
 * Is creep analysis to be considered?
 yes
 * Is shrink analysis to be considered?
 yes

CONCRETE LAYER SYSTEM CONTROL

* Number of concrete layers
 1 10

CONCRETE LAYER BOUNDARY COORDINATES

* Coordinates of the bottom of the slab
 -10.00
 * Boundary between layer 1 and 2
 -9.90
 * Boundary between layer 2 and 3
 -8.50
 * Boundary between layer 3 and 4
 -6.00
 * Boundary between layer 4 and 5
 -3.00
 * Boundary between layer 5 and 6
 .00
 * Boundary between layer 6 and 7
 3.00
 * Boundary between layer 7 and 8
 6.00
 * Boundary between layer 8 and 9
 8.50
 * Boundary between layer 9 and *
 9.90
 * Coordinates of the top of the slab
 10.00

CONCRETE LAYER REGION ASSIGNMENT

* Region	Layer Sys.
* Number	Number
* Span number	1
1	1
* Span number	2
1	1
* Span number	3
1	1

TEMPERATURE AND HUMIDITY

*
 * Initial Temperature
 70.00
 * Percentage relative ambient humidity
 60.50
 * Temperature Constant for all of analysis?
 yes

 * Reinforcing Steel

REINFORCING STEEL PROPERTIES
 * modulus of elasticity
 29000.000000
 * yield stress Fy
 60.00
 * modulus for strain-hardening
 346.00
 * ultimate strain
 1.800000E-01
 * number of steel layer systems
 1

*
 NUMBER OF REINFORCING STEEL LAYERS

* layer sys | number of
 * number | steel layers
 1 | 4

*
 REINFORCING STEEL LAYER DATA

layer number	z coord. to center of layer	steel number	spacing	angle (deg.) the direction of rebars
1	8.00	5	1.000	90.000
2	7.37	5	1.000	.000
3	-7.37	5	1.000	.000
4	-8.00	5	1.000	90.000

*
 REINFORCING STEEL LAYER REGION ASSIGNMENT

Region Number	Layer Sys. Number
Span number 1	1
1	1
Span number 2	2
1	1
Span number 3	3
1	1

 * Prestressing Steel

NUMBER OF PRESTRESSING STEEL TYPES

* number of prestressing steel types
 2

PRESTRESSING STEEL DATA

prestress steel material number	bond code*	strand area	number of strands	ultimate strength	frac. of ult.
1	1	.1530	19	270.00	.750
2	1	.1530	7	270.00	.750

* (Note: *-bond code: 0=post-tensioned unbonded, 1=post-tension bonded, 2=pretensioned)

prestress steel material number	wobble frict. coeff.	curvature friction coeff.	0.1% offset yield stress	relaxation coefficient	# of points on stress-strain curve
1	.00020	.250000	225.00	10.00	4
2	.00020	.250000	225.00	10.00	4

* stress-strain curve for material type # 1

```

* stress      strain
  170.520    .58000E-02
  220.000    .10000E-01
  240.000    .30000E-01
  253.000    .67000E-01

```

```

* stress-strain curve for material type # 2

```

```

* stress      strain
  170.520    .58000E-02
  220.000    .10000E-01
  240.000    .30000E-01
  253.000    .67000E-01

```

```

*-----
* Prestressing Tendons
*-----

```

PRESTRESSING TENDON DATA CONTROL

```

* number of tendon groups
  9

```

* tendon group number	number of tendons in group	tendon material number	number of inflexion points	tendon dir. code	spacing between tendons	spacing between tendons	firs tendons starting	first tendons ending
1	20	1	6	1	2.800	2.800	1.400	1.400
2	4	1	2	2	6.000	6.000	41.000	41.000
3	3	1	2	2	6.000	6.000	44.000	44.000
4	4	1	2	2	6.000	6.000	91.000	91.000
5	3	1	2	2	6.000	6.000	94.000	94.000
6	1	2	2	2	.000	.000	3.000	3.000
7	1	2	2	2	.000	.000	147.000	147.000
8	1	2	2	2	.000	.000	3.000	3.000
9	1	2	2	2	.000	.000	147.000	147.000

```

* Comments explaining the above. Change line # 680 in bwrite.for

```

PRESTRESSING TENDON JACKING FORCE DATA

* tendon group no.	anchor slip	factor for jacking force at start of tendon	factor for jacking force at end of tendon
1	5.000000E-01	1.000000	1.000000
2	5.000000E-01	0.000000E+00	1.000000
3	2.500000E-01	1.000000	0.000000E+00
4	5.000000E-01	0.000000E+00	1.000000
5	5.000000E-01	1.000000	0.000000E+00
6	5.000000E-01	0.000000E+00	1.000000
7	5.000000E-01	0.000000E+00	1.000000
8	2.500000E-01	1.000000	0.000000E+00
9	5.000000E-01	1.000000	0.000000E+00

INFLEXION POINT DATA

```

* tendon group # 1

```

* inflexion point number	inflexion point location	tendon eccentricity at the point	curvature code	distance to a point Q	tendon eccentricity at Q
1	0.000000E+00	0.000000E+00		0	1.344000E-01
2	3.023000E-01	3.500000		0	3.360000E-02
3	3.687000E-01	3.500000		0	1.313000E-01
4	6.313000E-01	3.500000		0	3.280000E-02
5	6.977000E-01	3.500000		0	1.680000E-01
6	1.000000	0.000000E+00		0	0.000000E+00

```

* tendon group # 2
*
* inflection|inflection| tendon|curvature|distance| tendon
* point    |point    |eccentricity|code|to a|eccentricity
* number   |location|at the point|   |point Q|at Q
*          |         |            |   |       |
1         |0.000000E+00| 0.000000E+00|   |0| 0.000000E+00
0.000000E+00
2         |1.000000| 0.000000E+00|   |0| 0.000000E+00
0.000000E+00
*
* tendon group # 3
*
* inflection|inflection| tendon|curvature|distance| tendon
* point    |point    |eccentricity|code|to a|eccentricity
* number   |location|at the point|   |point Q|at Q
*          |         |            |   |       |
1         |0.000000E+00| 0.000000E+00|   |0| 0.000000E+00
0.000000E+00
2         |1.000000| 0.000000E+00|   |0| 0.000000E+00
0.000000E+00
*
* tendon group # 4
*
* inflection|inflection| tendon|curvature|distance| tendon
* point    |point    |eccentricity|code|to a|eccentricity
* number   |location|at the point|   |point Q|at Q
*          |         |            |   |       |
1         |0.000000E+00| 0.000000E+00|   |0| 0.000000E+00
0.000000E+00
2         |1.000000| 0.000000E+00|   |0| 0.000000E+00
0.000000E+00
*
* tendon group # 5
*
* inflection|inflection| tendon|curvature|distance| tendon
* point    |point    |eccentricity|code|to a|eccentricity
* number   |location|at the point|   |point Q|at Q
*          |         |            |   |       |
1         |0.000000E+00| 0.000000E+00|   |0| 0.000000E+00
0.000000E+00
2         |1.000000| 0.000000E+00|   |0| 0.000000E+00
0.000000E+00
*
* tendon group # 6
*
* inflection|inflection| tendon|curvature|distance| tendon
* point    |point    |eccentricity|code|to a|eccentricity
* number   |location|at the point|   |point Q|at Q
*          |         |            |   |       |
1         |0.000000E+00| 5.000000|   |0| 0.000000E+00
0.000000E+00
2         |1.000000| 5.000000|   |0| 0.000000E+00
0.000000E+00
*
* tendon group # 7
*
* inflection|inflection| tendon|curvature|distance| tendon
* point    |point    |eccentricity|code|to a|eccentricity
* number   |location|at the point|   |point Q|at Q
*          |         |            |   |       |
1         |0.000000E+00| 5.000000|   |0| 0.000000E+00
0.000000E+00
2         |1.000000| 5.000000|   |0| 0.000000E+00
0.000000E+00
*
* tendon group # 8
*
* inflection|inflection| tendon|curvature|distance| tendon
* point    |point    |eccentricity|code|to a|eccentricity
* number   |location|at the point|   |point Q|at Q
*          |         |            |   |       |
1         |0.000000E+00| -5.000000|   |0| 0.000000E+00
0.000000E+00
2         |1.000000| -5.000000|   |0| 0.000000E+00
0.000000E+00
*
* tendon group # 9

```

```

*
* inflection|inflection| tendon|curvature|distance| tendon
* point| point| eccentricity| code| to a| eccentricity
* number| location| at the point| point Q| at Q
1 0.000000E+00 -5.000000 0 0.000000E+00
0.000000E+00
2 1.000000 -5.000000 0 0.000000E+00
0.000000E+00

```

```

*
*
* Some notes go here. Line 770 of bwrite.for
*
*-----

```

* Loads

```

*-----
*
LOADS
*Number of Time Steps :
*Number of times analysis is to be performed for different days - Max 10
3
*Number of Lanes :
*Number of Lanes for AASHTO Lane Loading - Max 10
0
*Number of Trucks :
*Number of AASHTO Trucks - Max 10
1
*Number of Concentrated Loads:
*Number of Individual Concentrated Load Locations - Max 10
0
*Number of Distributed Load Regions :
*ie. Overlay-Dist. over parts or all of bridge - Max 10
0
*Number of Line Loads :
*ie. Gaurdrail type loads-along boundaries or rows only - Max 10
0

```

DAYS

```

*
28 56 365

```

LOAD CONTROL

* Load control table

* Days	fraction of dead load	fraction of distributed surf. loads	fraction of prestressing load	fraction of elastic def. at transfer	No. L Load Steps	No. T Load Steps
28	1.000	.500	.000	.000	.000	1
56	.000	.000	.000	.000	.000	0
365	.000	.000	1.000	.000	.000	1

LANE LOADS

* Lane no.	Conc. Load (kips)	Dist. Load (ksf)	Init Row Num	Fin Row Num	Concen. X (1) (ft)	Concen. Y (1) (ft)	Concen. X(2) (ft)	Concen. Y (2) (ft)
------------	-------------------	------------------	--------------	-------------	--------------------	--------------------	-------------------	--------------------

TRUCK LOADS

* truck number	front wheel kips	middle wheel kips	rear wheel kips	dir.	x coord feet	y coord feet	truck width feet	front spac feet	rear spac feet
1	4.00	16.00	16.00	0	22.000	4.000	6.00	14.00	14.00

CONCENTRATED LOADS

* load	location X	location Y	X-direction load	mom. about x-axis	mom. about y-axis
--------	------------	------------	------------------	-------------------	-------------------

DISTRIBUTED LOADS

* Num	Magnitude load/area	First Element	Last Element	Element Step
-------	---------------------	---------------	--------------	--------------

LINE LOADS

* Num	Magnitude load/len.	First Node	Last Node
-------	---------------------	------------	-----------

```
*-----  
* Convergence Parameters  
*-----  
CONVERGENCE CRITERIA  
* Displacement Convergence (in):  
  1.000000E-02  
* Rotation Convergence (rad):  
  1.000000E-01  
UPPER LIMITS  
* Maximum Displacement (in):  
  5.000000  
* Maximum Rotation (rad):  
  5.000000E-02  
NONLINEAR CONTROL  
* Max Number of Iterations for Load Analysis:  
  10  
* Max Number of Iterations for Time Analysis:  
  10  
* Iteration Type Code:  
  0  
END OF FILE
```

Output Microfile - SAMPLE.OFL

```
*
* General Control
*
GENERAL OUTPUT CONTROL
*
* number of times analysis is performed
  4
*
* Specify if reactions are output for all analysis
yes
* Specify if reactions are output for last analysis only
no
*
* Specify graphical output
* Load Case No.  1
yes
* Load Case No.  2
no
* Load Case No.  3
no
* Load Case No.  4
yes
*
*-----
* Nodal Control
*-----
NODAL OUTPUT CONTROL
*
* time |
* step | output |
* number | control |
   1     2
   2     3
   3     3
   4     2
*
* Specify if displacements are output at all nodes
no
* Specify number of individual nodes
  0
* Specify number of group of series of nodes
  2
*
* Load Case No.
* | Node |
*
* Nodal Series Specification
* | Group | Beginning | Ending | Increment |
* | Number | Node | Node | Number |
   1     1   190     8
   2     5   194     8
*
* Specify if external forces at nodes
no
* Specify if unbalanced forces at nodes
no
*
* Specify if output displacements are to be displayed
yes
* Specify if displacements in local coordinates
no
*
*-----
* Element Control
*-----
ELEMENT OUTPUT CONTROL
*
```

```

* time |
* step | output |
* number | control |
      1 | 2 |
      2 | 3 |
      3 | 3 |
      4 | 2 |
*
* Specify if stresses are output at all layers
no
* Specify if stresses are output for t and b layers
yes
* Specify if stresses are output for t,m, and b layers
no
*
* Specify if output is for all layers
no
* Specify number of individual elements to output
10
* Specify number of element groups to output
0
*
* Individual Element Specification
* | Number | Element |
      1 | 1 |
      2 | 21 |
      3 | 55 |
      4 | 87 |
      5 | 104 |
      6 | 146 |
      7 | 193 |
      8 | 220 |
      9 | 268 |
     10 | 320 |
*
* Element Series Specification
* | Group | Beginning | Ending | Increment |
* | Number | Element | Element | Element |
*
* Specify if output concrete stresses
yes
* Specify if output concrete strains
yes
* Specify if principal stresses are output
yes
* Specify if concrete state codes are output
yes
* Specify if output concrete shrinkage strains
yes
* Specify if output concrete creep strains
yes
*
*-----
* Reinforcement/Prestress Control
*-----
REINFORCEMENTS/PRESTRESS OUTPUT
*
* Specify if output reinforcement stress
yes
* Specify if output stress reinforcement strains
no
*
* Specify if output prestress tendon geometry
yes
* Specify if output information for all tendons
no
* Specify if output tendons to be outputed
3
* Specify if tendon forces in elements are output
no
*

```

* Individual tendon specifications

* | Number | Tendon |

1 1

2 21

3 35

*

APPENDIX IV. EXAMPLE FEM INPUT DATA FILE

NOPARC Input Data File - SAMPLE.DAT

```

*-----
198  2  3  0  0  0  1  1  1  0  0
  2  0  0  1  0  2  4  1
  1  0  0  2  0  0  1  2  0  10  0  1  1  1  1  1  1
  1  0  0  1
  2  3  3  2
  2  3  3  2

  1 190  8  5 194  8
  1 21 55 87 104 146 193 220 268 320

  1 0 1 0 3 0
  1 21 35
  .0000 .0000 .0100 .0100
  .0000 .0000 5.0000 .0500
  28. 56. 365.

  1 0 0 0 0 0 1 .00 672.00 .00
  2 0 0 0 0 0 1 .00 588.00 .00
  3 0 0 0 0 0 1 .00 504.00 .00
  4 0 0 0 0 0 1 .00 420.00 .00
  5 0 0 0 0 0 1 .00 336.00 .00
  6 0 0 0 0 0 1 .00 252.00 .00
  7 0 0 0 0 0 1 .00 168.00 .00
  8 0 0 0 0 0 1 .00 84.00 .00
  9 0 0 0 0 0 1 .00 .00 .00
... etc.
190 0 0 0 0 0 1 1800.00 672.00 .00
191 0 0 0 0 0 1 1800.00 588.00 .00
192 0 0 0 0 0 1 1800.00 504.00 .00
193 0 0 0 0 0 1 1800.00 420.00 .00
194 0 0 0 0 0 1 1800.00 336.00 .00
195 0 0 0 0 0 1 1800.00 252.00 .00
196 0 0 0 0 0 1 1800.00 168.00 .00
197 0 0 0 0 0 1 1800.00 84.00 .00
198 0 0 0 0 0 1 1800.00 .00 .00
  1 1 2 1 1
  1 1 2 2 6300.0000 .1500 .0839 10.0000 1.0000
4589790. 6345. 621. .002765
  7.10000 20.00000 60.50000
  1 .2900E+08 .6000E+05 .3460E+06 .1800E+00
  1 1 .2907E+01 .1667E-04 .2500E+00 .2250E+06 .1000E+02 4
170520. .00580 220000. .01000 240000. .03000 253000. .06700
  2 1 .1071E+01 .1667E-04 .2500E+00 .2250E+06 .1000E+02 4
170520. .00580 220000. .01000 240000. .03000 253000. .06700
  1 10
-10.00000 -9.90000 -8.50000 -6.00000 -3.00000 .00000 3.00000 6.00000
  8.50000 9.90000 10.00000
  1 4 1
  1 1 8.00000 .31000 90.00000
  2 1 7.37000 .31000 .00000
  3 1 -7.37000 .31000 .00000
  4 1 -8.00000 .31000 90.00000
  1 336 0 0
  .00 .00 -1.00
  1 2 11 10 1 1 1 .0 .00 .00 .00 .00 70.00
  2 2 10 1 1 1 1 .0 .00 .00 .00 .00 70.00
  3 11 20 19 1 1 1 .0 .00 .00 .00 .00 70.00
  4 11 19 10 1 1 1 .0 .00 .00 .00 .00 70.00
  5 20 29 28 1 1 1 .0 .00 .00 .00 .00 70.00
  6 20 28 19 1 1 1 .0 .00 .00 .00 .00 70.00
  7 29 38 37 1 1 1 .0 .00 .00 .00 .00 70.00
... etc.
328 153 161 152 1 1 1 1 .0 .00 .00 .00 .00 70.00
329 162 171 170 1 1 1 1 .0 .00 .00 .00 .00 70.00

```

330	162	170	161	1	1	1	1	.0	.00	.00	.00	.00	70.00
331	171	180	179	1	1	1	1	.0	.00	.00	.00	.00	70.00
332	171	179	170	1	1	1	1	.0	.00	.00	.00	.00	70.00
333	180	189	188	1	1	1	1	.0	.00	.00	.00	.00	70.00
334	180	188	179	1	1	1	1	.0	.00	.00	.00	.00	70.00
335	189	198	197	1	1	1	1	.0	.00	.00	.00	.00	70.00
336	189	197	188	1	1	1	1	.0	.00	.00	.00	.00	70.00
2	60												
9	9	17	8	18	1	0	0	0.	0.	.51852E+06			
8	9	17	8	18	1	0	0	0.	0.	.51852E+06			
7	9	17	8	18	1	0	0	0.	0.	.51852E+06			
6	9	17	8	18	1	0	0	0.	0.	.51852E+06			
5	9	17	8	18	1	0	0	0.	0.	.51852E+06			
4	9	17	8	18	1	0	0	0.	0.	.51852E+06			
3	9	17	8	18	1	0	0	0.	0.	.51852E+06			
2	9	17	8	18	1	0	0	0.	0.	.51852E+06			
1	9	17	8	18	1	0	0	0.	0.	.51852E+06			
198	9	17	8	18	1	0	0	0.	0.	.51852E+06			
197	9	17	8	18	1	0	0	0.	0.	.51852E+06			
196	9	17	8	18	1	0	0	0.	0.	.51852E+06			
195	9	17	8	18	1	0	0	0.	0.	.51852E+06			
194	9	17	8	18	1	0	0	0.	0.	.51852E+06			
193	9	17	8	18	1	0	0	0.	0.	.51852E+06			
192	9	17	8	18	1	0	0	0.	0.	.51852E+06			
191	9	17	8	18	1	0	0	0.	0.	.51852E+06			
190	9	17	8	18	1	0	0	0.	0.	.51852E+06			
9	8	0	0	0	1	0	0	0.	0.	.14933E+05			
8	9	0	0	0	1	0	0	0.	0.	.14933E+05			
7	8	0	0	0	1	0	0	0.	0.	.14933E+05			
6	7	0	0	0	1	0	0	0.	0.	.14933E+05			
5	6	0	0	0	1	0	0	0.	0.	.14933E+05			
4	5	0	0	0	1	0	0	0.	0.	.14933E+05			
3	4	0	0	0	1	0	0	0.	0.	.14933E+05			
2	3	0	0	0	1	0	0	0.	0.	.14933E+05			
1	2	0	0	0	1	0	0	0.	0.	.14933E+05			
198	197	0	0	0	1	0	0	0.	0.	.14933E+05			
197	198	0	0	0	1	0	0	0.	0.	.14933E+05			
196	197	0	0	0	1	0	0	0.	0.	.14933E+05			
195	196	0	0	0	1	0	0	0.	0.	.14933E+05			
194	195	0	0	0	1	0	0	0.	0.	.14933E+05			
193	194	0	0	0	1	0	0	0.	0.	.14933E+05			
192	193	0	0	0	1	0	0	0.	0.	.14933E+05			
191	192	0	0	0	1	0	0	0.	0.	.14933E+05			
190	191	0	0	0	1	0	0	0.	0.	.14933E+05			
71	9	17	8	18	1	0	0	0.	0.	.85200E+07			
69	9	17	8	18	1	0	0	0.	0.	.85200E+07			
67	9	17	8	18	1	0	0	0.	0.	.85200E+07			
65	9	17	8	18	1	0	0	0.	0.	.85200E+07			
134	9	17	8	18	1	0	0	0.	0.	.85200E+07			
132	9	17	8	18	1	0	0	0.	0.	.85200E+07			
130	9	17	8	18	1	0	0	0.	0.	.85200E+07			
128	9	17	8	18	1	0	0	0.	0.	.85200E+07			
71	62	0	0	0	1	0	0	0.	0.	.42500E+05			
69	60	0	0	0	1	0	0	0.	0.	.42500E+05			
67	58	0	0	0	1	0	0	0.	0.	.42500E+05			
65	56	0	0	0	1	0	0	0.	0.	.42500E+05			
134	125	0	0	0	1	0	0	0.	0.	.42500E+05			
132	123	0	0	0	1	0	0	0.	0.	.42500E+05			
130	121	0	0	0	1	0	0	0.	0.	.42500E+05			
128	119	0	0	0	1	0	0	0.	0.	.42500E+05			
71	72	0	0	0	1	0	0	0.	0.	.72500E+05			
69	70	0	0	0	1	0	0	0.	0.	.72500E+05			
67	68	0	0	0	1	0	0	0.	0.	.72500E+05			
65	66	0	0	0	1	0	0	0.	0.	.72500E+05			
134	135	0	0	0	1	0	0	0.	0.	.72500E+05			
132	133	0	0	0	1	0	0	0.	0.	.72500E+05			
130	131	0	0	0	1	0	0	0.	0.	.72500E+05			
128	129	0	0	0	1	0	0	0.	0.	.72500E+05			
38	42	6	30										
1	0	0	1	42	6	9	8	198	197	-1	-1		
.50 588667.50-588667.50													

100	99	102	101	104	103	106	105	108	107	110	109	112	111	212	211
214	213	216	215	218	217	220	219	222	221	224	223	324	323	326	325
328	327	330	329	332	331	334	333	336	335						
	.00	16.80		.00		1.00		241.92		-6.00					
	544.14	16.80		3.50		1.00		60.48		5.50					
	663.66	16.80		3.50		1.00		236.34		-4.00					
	1136.34	16.80		3.50		1.00		59.04		5.50					
	1255.86	16.80		3.50		1.00		302.40		-6.00					
	1800.00	16.80		.00		1.00		.00		.00					
...	etc.														
22	0	0	1	16	2	63	72	55	64	-1	-1				
	.50		.00	-588667.50											
111	112	97	98	83	84	69	70	55	56	41	42	27	28	13	14
	564.00		.00	.00		.00		.00	.00	.00					
	564.00	672.00		.00		.00		.00	.00	.00					
23	0	0	1	16	2	72	81	64	73	-1	-1				
	.50		.00	-588667.50											
211	212	197	198	183	184	169	170	155	156	141	142	127	128	113	114
	636.00		.00	.00		.00		.00	.00	.00					
	636.00	672.00		.00		.00		.00	.00	.00					
24	0	0	1	16	2	81	90	73	82	-1	-1				
	.50		.00	-588667.50											
213	214	199	200	185	186	171	172	157	158	143	144	129	130	115	116
	708.00		.00	.00		.00		.00	.00	.00					
	708.00	672.00		.00		.00		.00	.00	.00					
25	0	0	1	16	2	63	72	55	64	-1	-1				
	.25	588667.50		.00											
111	112	97	98	83	84	69	70	55	56	41	42	27	28	13	14
	528.00		.00	.00		.00		.00	.00	.00					
	528.00	672.00		.00		.00		.00	.00	.00					
26	0	0	1	16	2	72	81	64	73	-1	-1				
	.25	588667.50		.00											
211	212	197	198	183	184	169	170	155	156	141	142	127	128	113	114
	600.10		.00	.00		.00		.00	.00	.00					
	600.10	672.00		.00		.00		.00	.00	.00					
27	0	0	1	16	2	72	81	64	73	-1	-1				
	.25	588667.50		.00											
211	212	197	198	183	184	169	170	155	156	141	142	127	128	113	114
	672.10		.00	.00		.00		.00	.00	.00					
	672.10	672.00		.00		.00		.00	.00	.00					
28	0	0	1	16	2	117	126	109	118	-1	-1				
	.50		.00	-588667.50											
221	222	207	208	193	194	179	180	165	166	151	152	137	138	123	124
	1092.00		.00	.00		.00		.00	.00	.00					
	1092.00	672.00		.00		.00		.00	.00	.00					
29	0	0	1	16	2	126	135	118	127	-1	-1				
	.50		.00	-588667.50											
223	224	209	210	195	196	181	182	167	168	153	154	139	140	125	126
	1164.00		.00	.00		.00		.00	.00	.00					
	1164.00	672.00		.00		.00		.00	.00	.00					
30	0	0	1	16	2	135	144	127	136	-1	-1				
	.50		.00	-588667.50											
323	324	309	310	295	296	281	282	267	268	253	254	239	240	225	226
	1236.00		.00	.00		.00		.00	.00	.00					
	1236.00	672.00		.00		.00		.00	.00	.00					
31	0	0	1	16	2	144	153	136	145	-1	-1				
	.50		.00	-588667.50											
325	326	311	312	297	298	283	284	269	270	255	256	241	242	227	228
	1308.00		.00	.00		.00		.00	.00	.00					
	1308.00	672.00		.00		.00		.00	.00	.00					
32	0	0	1	16	2	126	135	118	127	-1	-1				
	.50	588667.50		.00											
223	224	209	210	195	196	181	182	167	168	153	154	139	140	125	126
	1128.00		.00	.00		.00		.00	.00	.00					
	1128.00	672.00		.00		.00		.00	.00	.00					
33	0	0	1	16	2	135	144	127	136	-1	-1				
	.50	588667.50		.00											
323	324	309	310	295	296	281	282	267	268	253	254	239	240	225	226
	1200.10		.00	.00		.00		.00	.00	.00					
	1200.10	672.00		.00		.00		.00	.00	.00					
34	0	0	1	16	2	135	144	127	136	-1	-1				

APPENDIX V. EXAMPLE FEM OUTPUT DATA FILE

NOPARC Output Data File - SAMPLE.OUT

1*-----

NUMBER OF MODAL POINTS	198
NUMBER OF ELEMENT TYPES	2
NUMBER OF TIME STEPS	3
ITERATION TYPE CODE	0
<ul style="list-style-type: none"> -1 = INITIAL STIFFNESS ONLY 0 = CONSTANT STIFFNESS IN LOAD STEPS N = REFORM STIFFNESS EACH N ITERATIONS 	
CODE FOR NONLINEAR GEOMETRY	0
GEOMETRIC STIFFNESS CODE	0
<ul style="list-style-type: none"> 0 = NOT CONSIDERED 1 = INCLUDED 	
CREEP ANALYSIS CODE	1
SHRINKAGE ANALYSIS CODE	1
<ul style="list-style-type: none"> 0 = ANALYSIS NOT REQUIRED 1 = ANALYSIS REQUIRED 	
CONVERGENCE NORM CODE	1
<ul style="list-style-type: none"> 0 = FORCE NORM USED 1 = DISPLACEMENT NORM USED 2 = BOTH FORCE AND DISPL NORMS 	
CONVERGENCE TOLERANCE TYPE CODE	0
<ul style="list-style-type: none"> 0 = ABSOLUTE VALUES 1 = FRACTIONS 	
PRINCIPAL AXES DIRECTION CODE	0
<ul style="list-style-type: none"> 0 = CALCULATED IN PROGRAM 1 = COINCIDE WITH ELEMENT LOCAL AXES 	
OUTPUT CONTROL CODES	
<ul style="list-style-type: none"> 0 = NO 1 = YES 	
DISPL, UNBAL FORCES + STRESSES FOR EACH ITER	2
<ul style="list-style-type: none"> 2 = ONLY AT END OF TIME STEPS 	
MODAL DISPL IN LOCAL COORD SYSTEM	0
STRESS RESULTANTS	0
STRAINS	1
DISPL FOR EACH ITERATION	0
UNBAL FORCES FOR EACH ITERATION	2
CODE TO START STOP PRINTING OF ALGOR OUTPUT	4
CODE TO SUPPRESS STRAIN, STRESS, TENDON FORCE	1
TOLERANCES TO GET CONVERGENCE	
FORCES	.000000+00
MOMENTS	.000000+00
TRANSLATIONS	.100000-01
ROTATIONS	.100000-01

UPPER LIMITS ON UNBALANCE
 FORCES .000+00
 MOMENTS .000+00
 TRANSLATIONS .500+01
 ROTATIONS .500-01

ANALYSIS REQD. AT FOLLOWING DAYS AFTER CASTING
 28. 56. 365.
 1 COMPLETE MODAL POINT DATA

ONODE NUMBER	BOUNDARY CONDITION CODES						MODAL POINT COORDINATES		
	X	Y	Z	XX	YY	ZZ	X	Y	Z
1	0	0	0	0	0	1	0.0000+00	6.7200+02	0.0000+00
2	0	0	0	0	0	1	0.0000+00	5.8800+02	0.0000+00
3	0	0	0	0	0	1	0.0000+00	5.0400+02	0.0000+00
195	0	0	0	0	0	1	1.8000+03	2.5200+02	0.0000+00
196	0	0	0	0	0	1	1.8000+03	1.6800+02	0.0000+00
197	0	0	0	0	0	1	1.8000+03	8.4000+01	0.0000+00
198	0	0	0	0	0	1	1.8000+03	0.0000+00	0.0000+00

1 MATERIAL PROPERTIES - CONCRETE , REINFORCING STEEL AND PRESTRESSING STEEL

NUMBER OF CONCRETE TYPES 1
 NUMBER OF RE STEEL TYPES 1
 NUMBER OF PRE STEEL TYPES 2
 NUMBER OF CONCRETE LAYER SYSTEMS 1
 NUMBER OF RE STEEL LAYER SYSTEMS 1

CONCRETE MATERIAL PROPERTIES

TYPE NO. 1
 ELASTIC MATERIAL DATA INPUT INDICATOR 1
 CREEP DATA INPUT INDICATOR 2
 SHRINKAGE DATA INPUT INDICATOR 2
 DATA INPUT INDICATORS - 1 = READ IN VALUES
 2 = USE ACI DATA

COMPRESSIVE STRENGTH AT 28 DAYS .630000+04
 POISSONS RATIO .150000+00
 WEIGHT PER UNIT VOLUME .839000-01
 CRACKED SHEAR CONSTANT .100000+01

DAYS AFTER CASTING 28.
 COMPRESSIVE STRENGTH .634500+04
 TENSILE STRENGTH .621000+03
 MODULUS OF ELASTICITY .458980+07
 STRAIN AT COMPRESSIVE STRENGTH .276500-02
 ULTIMATE STRAIN IN COMPRESSION .110600-01
 ULTIMATE STRAIN IN TENSION .135300-02

TENSION STIFFENING MODEL - UNLOADING IN CONCRETE

ULTIMATE SHRINKAGE -.800000-03
 SLUMP OF MIX .710000+01
 SIZE OF MEMBER .200000+02
 RELATIVE HUMIDITY .605000+02
 TEMPERATURE COEFFICIENT .550000-05

STEEL MATERIAL PROPERTIES

TYPE	MODULUS	YIELD STRENGTH	BI-MODULUS	ULT STRAIN
1	.29000D+08	.60000D+05	.34600D+06	.18000D+00

PRESTRESSING STEEL PROPERTIES

BOND CODE 0 = POST-TENSIONED - UNBONDED
 1 = POST TENSIONED - BONDED
 2 = PRETENSIONED

TYPE NO	BOND CODE	AREA	WOBBLE COEF	FRICTION COEF	0.1 PERC. FY	RELAX COEF
1	1	2.907D+00	1.66700D-05	2.50000D-01	2.25000D+05	1.00D+01
2	1	1.071D+00	1.66700D-05	2.50000D-01	2.25000D+05	1.00D+01

POINTS ON THE STRESS-STRAIN CURVE - TYPE NO 1

SECTION	E-MODULUS	MAX STRESS	MAX STRAIN
1	2.94000D+07	1.70520D+05	5.80000D-03
2	1.17810D+07	2.20000D+05	1.00000D-02
3	1.00000D+06	2.40000D+05	3.00000D-02
4	3.51351D+05	2.53000D+05	6.70000D-02

POINTS ON THE STRESS-STRAIN CURVE - TYPE NO 2

SECTION	E-MODULUS	MAX STRESS	MAX STRAIN
1	2.94000D+07	1.70520D+05	5.80000D-03
2	1.17810D+07	2.20000D+05	1.00000D-02
3	1.00000D+06	2.40000D+05	3.00000D-02
4	3.51351D+05	2.53000D+05	6.70000D-02

1CONCRETE LAYER SYSTEMS

TYPE NO. 1
 Z-COORDINATES =
 -10.00000 -9.90000 -8.50000 -6.00000 -3.00000 .00000 3.00000 6.00000 8.50000 9.90000
 10.00000

STEEL LAYER SYSTEMS

TYPE NO. 1
 NO. OF LAYERS 4
 ANGLE CODE 1

LAYER	MATERIAL	Z-COORD.	SMEARED THK.	ANGLE
1	1	8.00000D+00	3.10000D-01	9.00000D+01
2	1	7.37000D+00	3.10000D-01	0.00000D+00
3	1	-7.37000D+00	3.10000D-01	0.00000D+00
4	1	-8.00000D+00	3.10000D-01	9.00000D+01

1TRIANGULAR SHELL ELEMENT DATA

NUMBER OF ELEMENTS 336
 ELEMENT TYPE OPTION 0
 0 = SHELL
 1 = MEMBRANE (CST)
 2 = PLATE BENDING (RAZZAQUE)
 OPTION FOR ELEMENT NODAL LOADS 0
 0 = CONSISTENT
 1 = TRIBUTARY AREA

GRAVITY LOAD MULTIPLIERS

X	Y	Z
.000	.000	-1.000

ELEMENT PY	NODE I PZ	NODE J TEMP	NODE K	CONCR	C L S	ST L S	LOCO	ANLO	PLAT	PX
1	2	11	10	1	1	1	1	.00	0.00000D+00	0.00000D+00
0.00000D+00	0.00000D+00	7.00000D+01								
2	2	10	1	1	1	1	1	.00	0.00000D+00	0.00000D+00
0.00000D+00	0.00000D+00	7.00000D+01								

3	11	20	19	1	1	1	1	.00	0.000000+00	0.000000+00
0.000000+00	0.000000+00	7.000000+01	7.000000+01	1	1	1	1	.00	0.000000+00	0.000000+00
333	180	189	188	1	1	1	1	.00	0.000000+00	0.000000+00
0.000000+00	0.000000+00	7.000000+01	7.000000+01	1	1	1	1	.00	0.000000+00	0.000000+00
334	180	188	179	1	1	1	1	.00	0.000000+00	0.000000+00
0.000000+00	0.000000+00	7.000000+01	7.000000+01	1	1	1	1	.00	0.000000+00	0.000000+00
335	189	198	197	1	1	1	1	.00	0.000000+00	0.000000+00
0.000000+00	0.000000+00	7.000000+01	7.000000+01	1	1	1	1	.00	0.000000+00	0.000000+00
336	189	197	188	1	1	1	1	.00	0.000000+00	0.000000+00
0.000000+00	0.000000+00	7.000000+01	7.000000+01							

... etc.

1 PRESTRESSING TENDON DATA

NUMBER OF TENDONS	38
MAX NO OF ELEMENTS CROSSED BY A TENDON	42
MAX NO OF INFLEXION POINTS PER TENDON	6
MAX NO OF TENDONS IN ONE ELEMENT	30

TENDON INFORMATION

TCODE - 0 = SLAB TENDON - IN ELEMENTS
 1 = SLAB TENDON - ON NODES
 2 = PANEL TENDON - IN ELEMENTS
 JCODE - 0 = JACKING FROM ONE END OR SEQUENTIAL
 1 = JACKING SYMMETRICALLY

TENDON NO	TCODE	JCODE	TYPE	NO EL	NO I P	NODE A	NODE B	NODE Y	NODE Z	ANCH SLIP
FORCE JS	FORCE JE									
1	0	0	1	42	6	9	8	198	197	5.000000-01
5.890+05	-5.890+05									
21	0	0	1	16	2	54	63	46	55	5.000000-01
0.000+00	-5.890+05									
35	0	0	2	16	2	9	18	1	10	5.000000-01
0.000+00	-2.170+05									

TENDON NO 1 (ANCHOR SLIP DISTANCE = .870+03)

1	0.00000+00	0.00000+00	4.91590+05	0.00000+00	-4.98030-02	0.00000+00				9
8	8.00000-01	2.00000-01								
2	1.71420+01	3.54320-03	4.92170+05	-8.23360-01	-4.62600-02	0.00000+00				9
17	8.00000-01	2.00000-01								
3	8.57100+01	1.77160-02	4.94480+05	-3.50940+00	-3.20880-02	0.00000+00				17
18	2.00000-01	8.00000-01								
4	1.02850+02	2.12590-02	4.95060+05	-4.02920+00	-2.85440-02	0.00000+00				18
26	8.00000-01	2.00000-01								
5	1.71430+02	3.54340-02	4.97390+05	-5.50060+00	-1.43700-02	0.00000+00				26
27	2.00000-01	8.00000-01								
6	1.88570+02	3.89770-02	4.97970+05	-5.71660+00	-1.08260-02	0.00000+00				27
35	8.00000-01	2.00000-01								
7	2.57140+02	5.31500-02	5.00310+05	-5.97300+00	3.34620-03	0.00000+00				35
36	2.00000-01	8.00000-01								
8	2.74280+02	5.66930-02	5.00900+05	-5.88530+00	6.88980-03	0.00000+00				36
44	8.00000-01	2.00000-01								
9	3.42860+02	7.08680-02	5.03250+05	-4.92680+00	2.10640-02	0.00000+00				44
45	2.00000-01	8.00000-01								
10	3.60000+02	7.44110-02	5.03840+05	-4.53530+00	2.46070-02	0.00000+00				45
53	8.00000-01	2.00000-01								
11	4.28570+02	8.85830-02	5.06210+05	-2.36220+00	3.87800-02	0.00000+00				53
54	2.00000-01	8.00000-01								
12	4.45710+02	9.21270-02	5.06800+05	-1.66690+00	4.23240-02	0.00000+00				54
62	8.00000-01	2.00000-01								
13	5.14290+02	1.06300-01	5.09180+05	1.72150+00	5.64980-02	0.00000+00				62
63	2.00000-01	8.00000-01								
14	5.31430+02	1.09840-01	5.09780+05	2.72030+00	6.00410-02	0.00000+00				63
71	8.00000-01	2.00000-01								
15	6.00000+02	1.70770-01	5.18190+05	5.49180+00	4.36880-03	0.00000+00				71
72	2.00000-01	8.00000-01								

	16	6.1714D+02	1.8997D-01	5.2083D+05	5.4021D+00	-1.4834D-02	0.0000D+00	72
80	8.0000D-01	2.0000D-01						
	17	6.8571D+02	2.5148D-01	5.2951D+05	2.1658D+00	-5.7546D-02	0.0000D+00	80
81	2.0000D-01	8.0000D-01						
	18	7.0285D+02	2.5608D-01	5.3027D+05	1.2187D+00	-5.2943D-02	0.0000D+00	81
89	8.0000D-01	2.0000D-01						
	19	7.7143D+02	2.7450D-01	5.3333D+05	-1.7804D+00	-3.4527D-02	0.0000D+00	89
90	2.0000D-01	8.0000D-01						
	20	7.8857D+02	2.7910D-01	5.3409D+05	-2.3328D+00	-2.9923D-02	0.0000D+00	90
98	8.0000D-01	2.0000D-01						
	21	8.5714D+02	2.9752D-01	5.3717D+05	-3.7533D+00	-1.1510D-02	0.0000D+00	98
99	2.0000D-01	8.0000D-01						
	22	8.7428D+02	3.0212D-01	5.3795D+05	-3.9112D+00	-6.9059D-03	0.0000D+00	99
107	8.0000D-01	2.0000D-01						
	23	9.4286D+02	3.2054D-01	5.3697D+05	-3.7533D+00	1.1510D-02	0.0000D+00	107
108	2.0000D-01	8.0000D-01						
	24	9.6000D+02	3.2514D-01	5.3620D+05	-3.5166D+00	1.6113D-02	0.0000D+00	108
116	8.0000D-01	2.0000D-01						
	25	1.0286D+03	3.4355D-01	5.3312D+05	-1.7804D+00	3.4527D-02	0.0000D+00	116
117	2.0000D-01	8.0000D-01						
	26	1.0457D+03	3.4816D-01	5.3236D+05	-1.1491D+00	3.9131D-02	0.0000D+00	117
125	8.0000D-01	2.0000D-01						
	27	1.1143D+03	3.6657D-01	5.2931D+05	2.1658D+00	5.7546D-02	0.0000D+00	125
126	2.0000D-01	8.0000D-01						
	28	1.1314D+03	3.7118D-01	5.2855D+05	3.1917D+00	6.2150D-02	0.0000D+00	126
134	8.0000D-01	2.0000D-01						
	29	1.2000D+03	4.4033D-01	5.1889D+05	5.4918D+00	-4.3688D-03	0.0000D+00	134
135	2.0000D-01	8.0000D-01						
	30	1.2171D+03	4.5953D-01	5.1626D+05	5.2523D+00	-2.3572D-02	0.0000D+00	135
143	8.0000D-01	2.0000D-01						
	31	1.2857D+03	5.1335D-01	5.0878D+05	1.7215D+00	-5.6498D-02	0.0000D+00	143
144	2.0000D-01	8.0000D-01						
	32	1.3029D+03	5.1690D-01	5.0818D+05	7.8324D-01	-5.2954D-02	0.0000D+00	144
152	8.0000D-01	2.0000D-01						
	33	1.3714D+03	5.3107D-01	5.0581D+05	-2.3621D+00	-3.8780D-02	0.0000D+00	152
153	2.0000D-01	8.0000D-01						
	34	1.3886D+03	5.3461D-01	5.0522D+05	-2.9965D+00	-3.5237D-02	0.0000D+00	153
161	8.0000D-01	2.0000D-01						
	35	1.4571D+03	5.4879D-01	5.0285D+05	-4.9268D+00	-2.1064D-02	0.0000D+00	161
162	2.0000D-01	8.0000D-01						
	36	1.4743D+03	5.5233D-01	5.0227D+05	-5.2575D+00	-1.7521D-02	0.0000D+00	162
170	8.0000D-01	2.0000D-01						
	37	1.5429D+03	5.6651D-01	4.9992D+05	-5.9730D+00	-3.3462D-03	0.0000D+00	170
171	2.0000D-01	8.0000D-01						
	38	1.5600D+03	5.7005D-01	4.9933D+05	-6.0000D+00	1.9694D-04	0.0000D+00	171
179	8.0000D-01	2.0000D-01						
	39	1.6286D+03	5.8422D-01	4.9700D+05	-5.5006D+00	1.4370D-02	0.0000D+00	179
180	2.0000D-01	8.0000D-01						
	40	1.6457D+03	5.8776D-01	4.9641D+05	-5.2238D+00	1.7913D-02	0.0000D+00	180
188	8.0000D-01	2.0000D-01						
	41	1.7143D+03	6.0194D-01	4.9409D+05	-3.5094D+00	3.2087D-02	0.0000D+00	188
189	2.0000D-01	8.0000D-01						
	42	1.7314D+03	6.0548D-01	4.9352D+05	-2.9290D+00	3.5631D-02	0.0000D+00	189
197	8.0000D-01	2.0000D-01						
	43	1.8000D+03	6.1965D-01	4.9121D+05	-8.6320D-15	4.9803D-02	0.0000D+00	197
198	2.0000D-01	8.0000D-01						

... etc.

NUMBER OF EQUATIONS 990
BANDWIDTH 50

1LOAD CONTROL DATA

NUMBER OF LOAD STEPS 1
NUMBER OF ITERATIONS PERMITTED 10

NUMBER OF LOADED JOINTS 0
 FRACTION OF DEAD LOAD .10000+01
 FRACTION OF SURFACE LOAD .00000+00
 FRACTION OF SPRING LOAD .00000+00
 FRACTION OF PRESTRESS LOAD .10000+01
 PRESTRESS - FRACTION OF EL DEF ALLOWED .50000+00
 NUMBER OF LOAD STEPS FOR TIME DEP. ANAL. 1
 NUMBER OF ITERATIONS FOR TIME DEP. ANAL. 10
 ITERATION TYPE CODE 0
 NUMBER OF ELEMENTS WITH TEMP CHANGE 0

ELEMENT AND TOTAL STIFFNESS MATRICES FORMED AND TRIANGULARIZED
 TIME STEP NO 1 LOAD STEP NO 1 ITERATION NO 1

... etc.

ELEMENT AND TOTAL STIFFNESS MATRICES FORMED AND TRIANGULARIZED
 TIME STEP NO 3 LOAD STEP NO 1 ITERATION NO 1

1*-----

==== RESULTS

TIME STEP NUMBER 3
 LOAD STEP NUMBER 1
 ITERATION NUMBER 1

JOINT DISPLACEMENTS

NODE	DISPL-X	DISPL-Y	DISPL-Z	ROTAT-X	ROTAT-Y	ROTAT-Z
1	3.889040-01	-6.129130-02	-4.501800-02	1.790880-04	-2.216130-03	0.000000+00
5	3.883970-01	6.582840-04	-6.942610-02	-2.118620-05	-2.372330-03	0.000000+00
9	3.912990-01	6.083860-02	-4.149480-02	-2.606240-04	-2.312700-03	0.000000+00
13	3.527210-01	-1.352320-02	1.228720-01	-1.961590-05	-2.046050-03	0.000000+00
17	3.555570-01	4.356970-02	1.305410-01	-1.240750-04	-1.986430-03	0.000000+00
21	3.174830-01	-2.863490-02	2.606450-01	-1.480850-05	-1.145390-03	0.000000+00
25	3.172140-01	2.994040-02	2.614320-01	-7.020430-06	-1.115120-03	0.000000+00
29	2.843630-01	-4.464660-02	3.085800-01	-2.733850-05	-3.674480-06	0.000000+00
33	2.772690-01	1.603110-02	3.096910-01	2.105510-05	2.073800-05	0.000000+00
37	2.579000-01	-6.410780-02	2.615600-01	-3.443070-05	9.975680-04	0.000000+00
41	2.382850-01	-6.344090-04	2.649330-01	4.600950-05	1.028490-03	0.000000+00
45	2.553110-01	6.595850-02	2.459550-01	1.588960-04	1.152300-03	0.000000+00
49	2.012260-01	-2.152170-02	1.544590-01	6.192590-05	1.541000-03	0.000000+00
53	2.102720-01	6.006340-02	1.423700-01	1.354630-04	1.512810-03	0.000000+00
57	1.663130-01	-4.269860-02	3.532960-02	6.289920-05	1.230280-03	0.000000+00
61	1.678870-01	4.098810-02	2.774130-02	4.874740-05	1.160590-03	0.000000+00
65	1.316250-01	-6.673090-02	-2.216740-02	-3.070820-05	1.346550-04	0.000000+00
69	1.323300-01	2.002910-02	-2.074990-02	-2.558810-07	-5.973910-06	0.000000+00
73	8.481200-02	-9.135410-02	6.329000-03	-3.275010-04	-1.119150-03	0.000000+00
77	9.681780-02	-4.367360-05	2.783910-02	-6.648480-05	-1.175090-03	0.000000+00
81	9.183120-02	9.129080-02	2.301350-02	1.916250-04	-1.290350-03	0.000000+00
85	5.927080-02	-2.195240-02	1.417380-01	-4.476120-05	-1.419940-03	0.000000+00
89	5.833320-02	6.454540-02	1.513430-01	2.238140-05	-1.331100-03	0.000000+00
93	1.910930-02	-4.080160-02	2.321060-01	-5.180060-05	-6.561270-04	0.000000+00
97	2.108000-02	4.172040-02	2.371980-01	-3.040030-06	-5.342590-04	0.000000+00
101	-1.877160-02	-5.806300-02	2.321170-01	-9.809270-05	4.820420-04	0.000000+00
105	-1.921800-02	2.086780-02	2.334910-01	1.642020-05	6.407650-04	0.000000+00
109	-4.748140-02	-7.886470-02	1.423760-01	-7.841860-05	1.333480-03	0.000000+00
113	-5.907290-02	-5.923930-04	1.443450-01	8.002710-05	1.419770-03	0.000000+00
117	-4.896310-02	8.091560-02	1.194140-01	3.025310-04	1.400110-03	0.000000+00
121	-9.756150-02	-2.143680-02	3.171420-02	6.415540-05	1.164240-03	0.000000+00
125	-9.499610-02	6.362410-02	2.260950-02	1.137050-04	1.102400-03	0.000000+00
129	-1.336310-01	-4.126460-02	-2.299700-02	4.214370-06	4.691640-05	0.000000+00
133	-1.326590-01	4.109210-02	-2.330620-02	1.443930-06	-8.812440-05	0.000000+00
137	-1.727030-01	-6.260660-02	2.405130-02	-1.148730-04	-1.112390-03	0.000000+00
141	-1.663540-01	2.045840-02	3.295650-02	-4.462500-05	-1.158880-03	0.000000+00
145	-2.209140-01	-7.848000-02	1.261690-01	-3.059200-04	-1.572780-03	0.000000+00

149	-2.01758D-01	2.49659D-04	1.50192D-01	-6.79119D-05	-1.54074D-03	0.00000D+00
153	-2.22016D-01	7.58483D-02	1.49687D-01	8.12773D-05	-1.54450D-03	0.00000D+00
157	-2.39704D-01	-1.79897D-02	2.61056D-01	-3.82349D-05	-1.03465D-03	0.00000D+00
161	-2.47302D-01	5.10262D-02	2.64798D-01	3.42355D-05	-9.70521D-04	0.00000D+00
165	-2.79955D-01	-3.14285D-02	3.06374D-01	-2.90696D-05	-3.56573D-05	0.00000D+00
169	-2.79534D-01	3.10769D-02	3.09082D-01	1.08396D-05	1.98293D-05	0.00000D+00
173	-3.19261D-01	-4.34298D-02	2.61898D-01	5.15595D-05	1.05904D-03	0.00000D+00
177	-3.15024D-01	1.45069D-02	2.59954D-01	1.33985D-05	1.14533D-03	0.00000D+00
181	-3.54732D-01	-5.78271D-02	1.42709D-01	2.21680D-04	1.90609D-03	0.00000D+00
185	-3.51870D-01	-4.70005D-04	1.22169D-01	1.85454D-05	2.03978D-03	0.00000D+00
189	-3.54606D-01	5.72146D-02	1.35621D-01	-1.49378D-04	1.91512D-03	0.00000D+00

CONCRETE LAYERS

NO. STRESS-XX STRESS-YY STRESS-XY STRESS-11 STRESS-22 STRESS-12 ANGLE PROJ. CODE 12 LR 12

...etc.

NODE	EXTENSIONAL STRESS	ROTATIONAL STRESS
9	2.15158735D+04	0.00000000D+00
8	3.00436358D+04	0.00000000D+00
7	3.43381284D+04	0.00000000D+00
6	3.58262497D+04	0.00000000D+00
5	3.59988068D+04	0.00000000D+00
4	3.58302957D+04	0.00000000D+00
3	3.46902711D+04	0.00000000D+00
2	3.09694161D+04	0.00000000D+00
1	2.33427431D+04	0.00000000D+00
198	2.30494283D+04	0.00000000D+00
197	3.10932292D+04	0.00000000D+00
196	3.48270757D+04	0.00000000D+00
195	3.59312418D+04	0.00000000D+00
194	3.60739997D+04	0.00000000D+00
193	3.58953721D+04	0.00000000D+00
192	3.44113007D+04	0.00000000D+00
191	3.00924635D+04	0.00000000D+00
190	2.13375964D+04	0.00000000D+00
9	-9.08503439D+02	0.00000000D+00
8	6.47679426D+02	0.00000000D+00
7	4.11867002D+02	0.00000000D+00
6	2.06762218D+02	0.00000000D+00
5	9.83016192D+00	0.00000000D+00
4	-1.81426751D+02	0.00000000D+00
3	-3.97763958D+02	0.00000000D+00
2	-6.30802418D+02	0.00000000D+00
1	-9.15262664D+02	0.00000000D+00
198	-9.13518680D+02	0.00000000D+00
197	6.31200859D+02	0.00000000D+00
196	3.99218024D+02	0.00000000D+00
195	1.83512373D+02	0.00000000D+00
194	-7.70347094D+00	0.00000000D+00
193	-2.04449556D+02	0.00000000D+00
192	-4.08989533D+02	0.00000000D+00
191	-6.43386564D+02	0.00000000D+00
190	-9.02310469D+02	0.00000000D+00
71	1.89697394D+05	0.00000000D+00
69	1.76789244D+05	0.00000000D+00
67	1.77101998D+05	0.00000000D+00
65	1.88866561D+05	0.00000000D+00
134	1.88543279D+05	0.00000000D+00
132	1.77245996D+05	0.00000000D+00
130	1.76806067D+05	0.00000000D+00
128	1.89391135D+05	0.00000000D+00
71	5.78043143D+03	0.00000000D+00
69	5.62400720D+03	0.00000000D+00
67	5.62993880D+03	0.00000000D+00
65	5.59407659D+03	0.00000000D+00
134	-5.59709701D+03	0.00000000D+00
132	-5.62905615D+03	0.00000000D+00
130	-5.62280597D+03	0.00000000D+00
128	-5.77949490D+03	0.00000000D+00

71	4.83686351D+03	0.00000000+00
69	1.45210955D+03	0.00000000+00
67	-1.45597109D+03	0.00000000+00
65	-4.83799384D+03	0.00000000+00
134	4.77935860D+03	0.00000000+00
132	1.43764227D+03	0.00000000+00
130	-1.44538457D+03	0.00000000+00
128	-4.78662122D+03	0.00000000+00

REFERENCES

- ALGOR ViziCad, modeling and visualization, volume 1.* (1990). Algor Interactive Systems, Inc., Pittsburgh, PA.
- ALGOR ViziCad plus, modeling and visualization, volume 2.* (1990). Algor Interactive Systems, Inc., Pittsburgh, PA.
- Building code requirements for reinforced concrete.* (1989). Amer. Concr. Inst., ACI Committee 318, Detroit, MI.
- Operating System/2™ standard edition, version 1.1.* (1988). International Business Machines Corporation, Armonk, NY.
- Operating System/2™ standard edition, version 1.3.* (1991). International Business Machines Corporation, Armonk, NY.
- Post-tensioning manual, fourth edition.* (1985). Post-Tensioning Institute, Phoenix, AZ., 149-160, 303-304.
- Prediction of creep, shrinkage and temperature effects in concrete structures.* (1970). Paper SP 27-3, Amer. Concr. Inst., ACI Committee 209, Detroit, MI.
- Roschke, P. N., and Inoue, M. (1991). "Effects of banded post-tensioning in prestressed concrete flat slab." *Report No. FHWA/TX-90/1182-1*, Texas Transportation Institute, Texas A&M Univ., College Station, TX.
- Roschke, P. N., Pruski, K. R., and Smith, C. D. (1992). "Experimental and analytical study of a two-span post-tensioned bridge slab." *Report No. FHWA/TX-92/1182-2*, Texas Transportation Institute, Texas A&M Univ., College Station, TX.
- Roschke, P. N., Pruski, K. R., and Sripadanna, N. (1992). "Experimental and analytical study of a post-tensioned bridge." *Report No. FHWA/TX-92/1182-3*, Texas Transportation Institute, Texas A&M Univ., College Station, TX.
- Standard specifications for highway bridges, thirteenth edition.* (1989). American Association of State Highway and Transportation Officials, Washington, D.C.
- van Greunen, J. (1979). "Nonlinear geometric, material and time dependent analysis of reinforced and prestressed concrete slabs and panels." *Research report No. UC SESM 79-3*, University of California, Berkeley, CA.

