Jurnal Teknologi, 39(B) Dis. 2003: 29–46

GENERATOR LOADS STUDY OF A HYDRO-ELECTRIC POWER HOUSE STRUCTURE

JAHANGIR BAKHTERI*

Abstract. The shape of a hydro-electric (hydel) power station particularly the concrete around spiral case and generator foundation is so complex that the determination of actual stresses is very complicated and time-consuming process since it is essentially a three-dimensional problem. The stress variations in the structure have been studied using three-dimensional photoelastic method in few small scale models only. In the present study, finite element and boundary element analyses of spiral (scroll) case and generator foundation of a large hydel power station considering generator loads have been carried out. A simple and efficient method of slices for the geometric representation of the structure has been developed. Linear elastic analyses have been carried out using well established existing finite element and boundary element computer programs and two pre-processing computer programs have also been developed to support the 3-D finite element program. Almost all of the present analyses show the same pattern of stresses and only their values differ, while most of the existing photoelastic study results indicate stresses with different pattern. In conclusion, a suitable, efficient and appropriate numerical method for the approximate analysis of the structure has been proposed.

Keywords: Scroll case, generator foundation, slicing technique, stresses

1.0 INTRODUCTION

A hydro-electric power station is either an underground power station or surface power station. A surface power station has three main subdivisions which are substructure, intermediate structure and superstructure. Intermediate structure is one of the most important parts of a power house structure which includes scroll case and generator foundation (Figure 1). The important function of the scroll case is to distribute water from penstock uniformly and smoothly through the guide vanes of the turbine and the generator foundation supports the generator [5,8]. A scroll case may be concrete or steel scroll case. In most construction cases, the steel scroll case is insulated from concrete around it and the present investigation has been carried out considering the same construction method [2,6].

In spite of the existence of the most sophisticated computer methods such as the finite element method (FEM) and boundary element method (BEM), no attempt has been made to study the stress concentration in this complicated structure by the above mentioned numerical methods. The design of the structure is usually based on

* Department of Structures & Materials, Faculty of Civil Engineering, University Teknologi Malaysia, 81310 Skudai, Johor, Malaysia, Tel: 607-5531706, Fax: 607-5566157, e-mail: jahangir@fka.utm.my

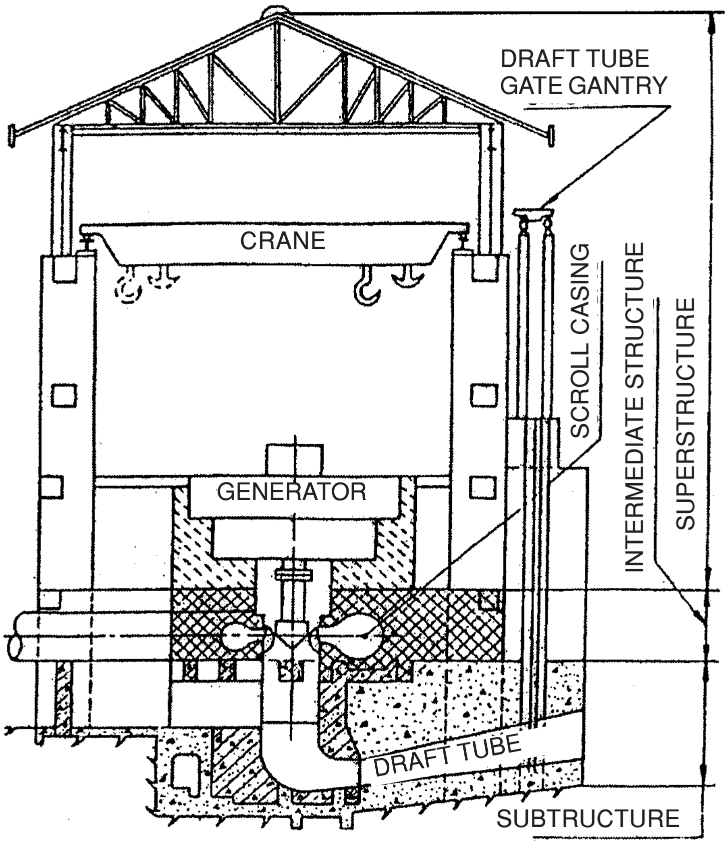


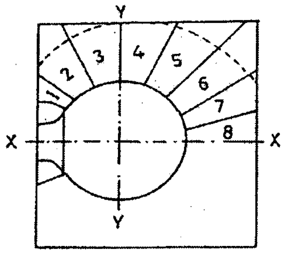
Figure 1 Typical section of a hydroelectric power stataion with vertical shaft reaction turbine showing the components

engineering judgement rather than on rational theory. Design organisations use different practices and a vertical section of the concrete around scroll case may be assumed as an arch arrangement or beam-column arrangement, while few organisations deal it as a ring arrangement (Figure 2). In some parts of the world it is considered as a mass concrete and only nominal reinforcement is provided against surface cracks and openings and to distribute shrinkage strains [4,6,8]. Photoelastic experimental stress analysis is one of the other important methods for stress analysis of the structure and Nigam (1972) has carried out photoelastic studies on few small scale power house models [6].

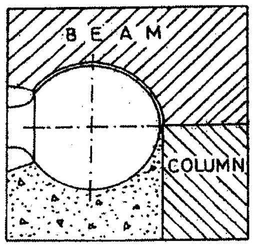
2.0 CASE STUDIES

2.1 Description of the Problem

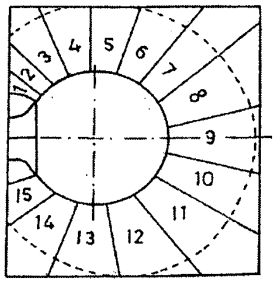
In order to study the validity of the available 3-D photoelastic analysis (i.e. the stress concentration factors (SCF)) given by Nigam, his original model (Model no.37) with



(a) Arch arrangement



(b) Beam-column arrangement



(c) Ring arrangement

Figure 2 Conventional method of analysis

its corresponding loads has been analyzed. Similarly, in order to know the actual stress concentration in the structure, a large hydel power station has been selected for the analysis. The power house has three units, each one of them with 24 MW power generation capacity. The scroll case has 3.4 m diameter at inlet (Figure 3).

The stress concentration in the power house structure has been modeled using Nigam's SCF as well as finite element and boundary element methods.

2.2 Loads and Boundary Conditions

The structure has been analysed under the action of generator loads. These loads have been simplified into UDL in the form of a circular ring with a constant width in plan. The stay vanes do not carry any loads and all the loads are transmitted through concrete at the side wall of the scroll case. The applied loads in the case of Nigam's model are included in Figure 4. Similarly the simplified generator loads acting on the structure are shown in Figure 5. The structure and the model are assumed to be supported on a rigid foundation at the bottom. For a proper assessment of stress concentrations in the power house, the structure in the analysis is represented at the bottom up to the top of the elbow portion of the draft tube and at the top up to the generator floor level.

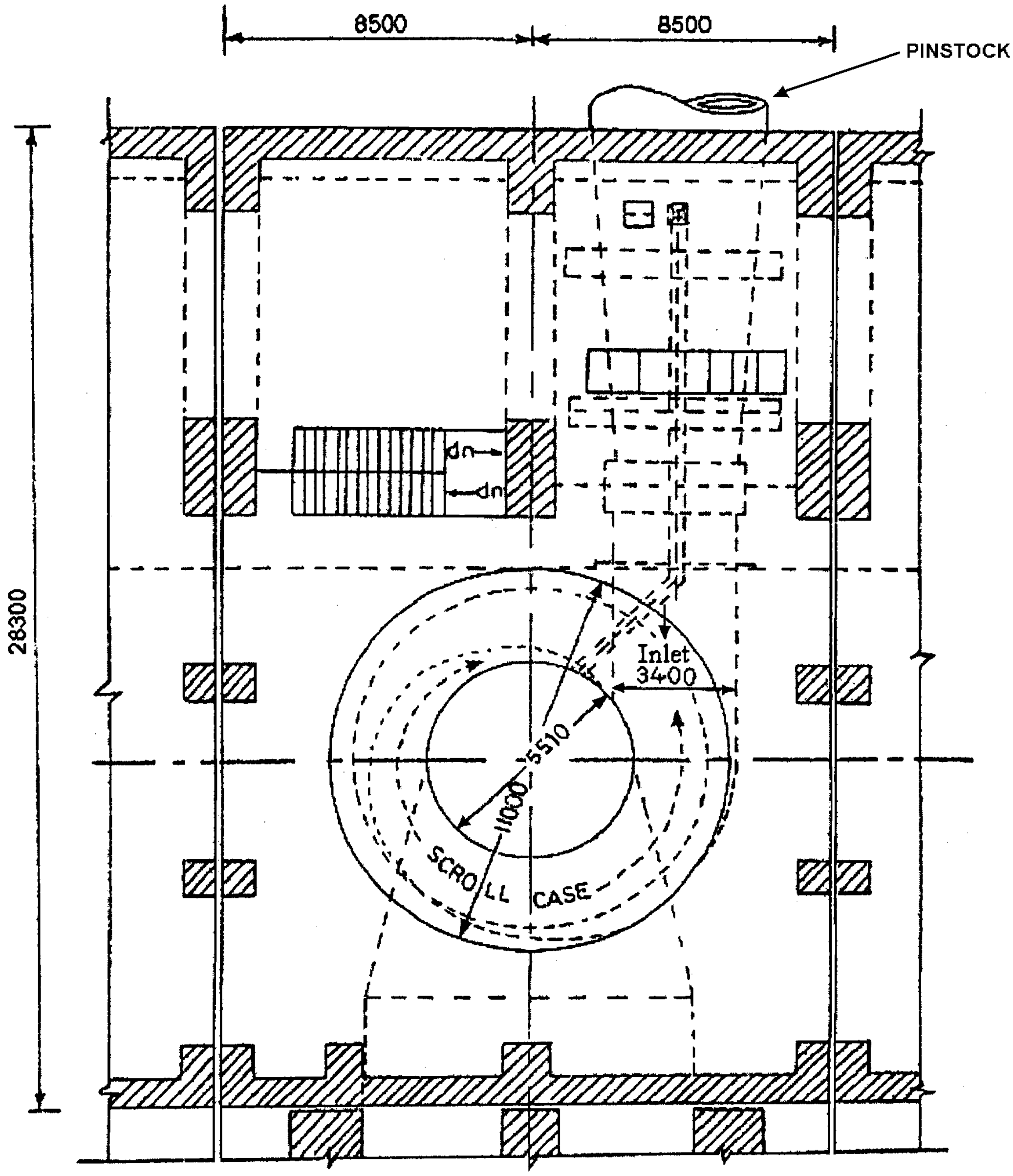


Figure 3 Plan of the power house showing the location of scroll case

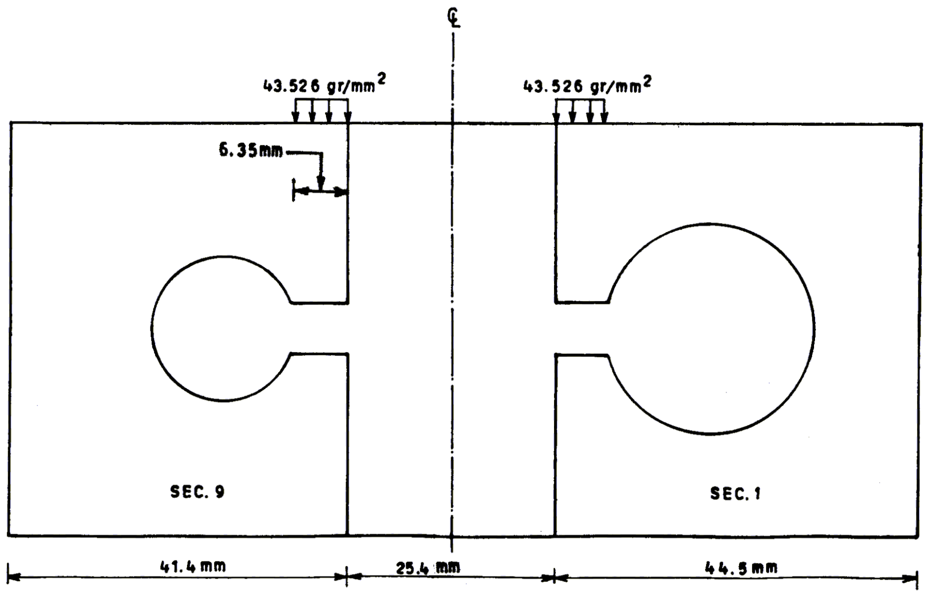


Figure 4 Longitudinal section of Nigam's model showing the position of simplified loads

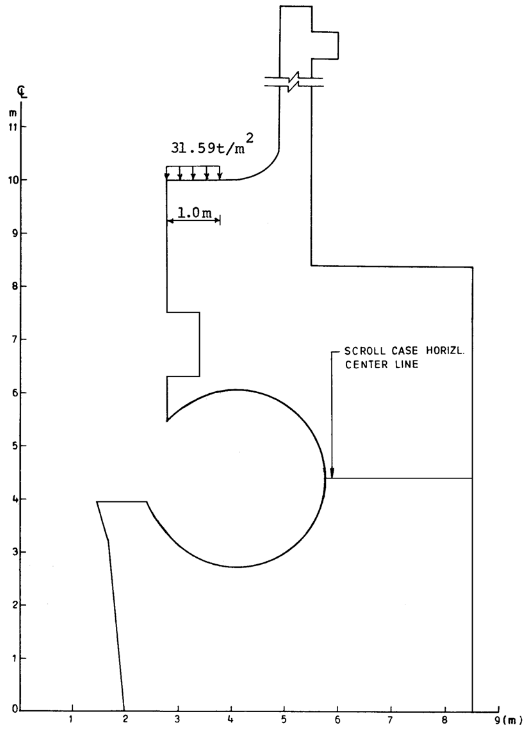


Figure 5 Power house cross-section at the inlet, showing the position of the simplified generator loads

3.0 IDEALIZATION BY METHOD OF SLICES

3.1 Slicing Technique

Due to the irregular cross-section and complicated cavities in a power house structure, its structural idealisation and analysis become very complicated for the structural design engineers. Therefore, an efficient and simple slicing method for the structural representation of the power house has been developed. This method which divides the structure into radial slices in plan, makes the comparison of the results from different analyses much easier. This study is a part of a detailed research which considers different load combinations. Most of the loads are transmitted to the structure through eight equally spaced pedestals [8]. In order to get an appropriate approximate stress concentration in the structure and the model, they have been divided into sixteen slices in 2-D analysis (Figure 6 and Figure 7). For 3-D finite element analyses the structure and the model have been divided into eight slices. However, due to the complicated cavities at the inlet part of the scroll case, slice no. 8 has further been divided into two slices which practically makes the structure and the model to be represented by nine radial slices (Figure 6 and Figure 7). The representation of the structure and the model by nine slices in 3-D is to ease the pre-processing and post-processing of the analysis.

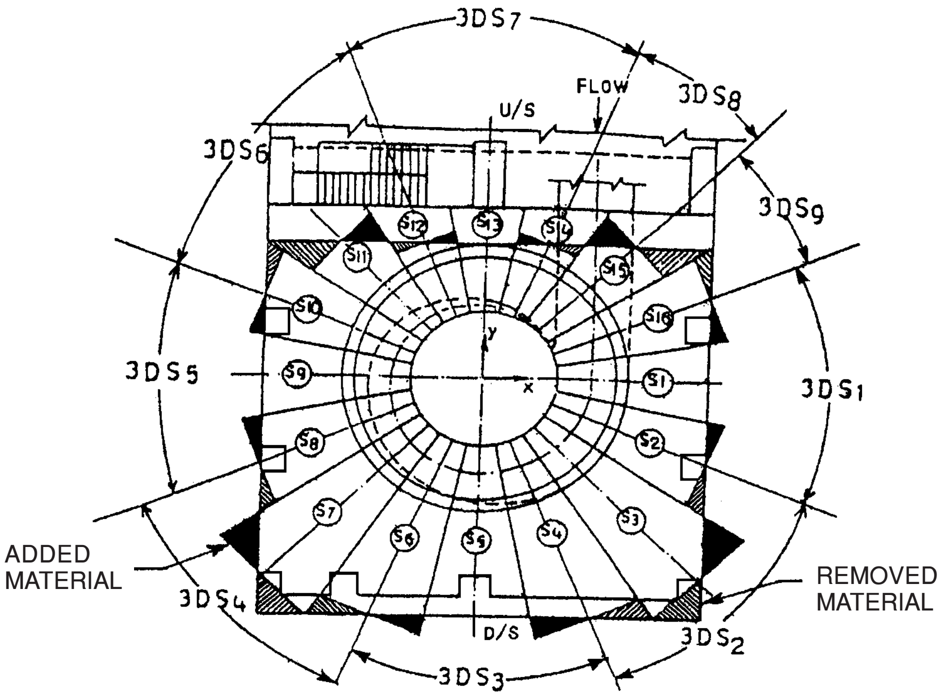


Figure 6 Power house in plan divided into sixteen slices in 2-D and nine slices in 3-D ($S_1 - S_{16}$ represent slices in 2-D and $3DS_1 - 3DS_9$ represent slices in 3-D)

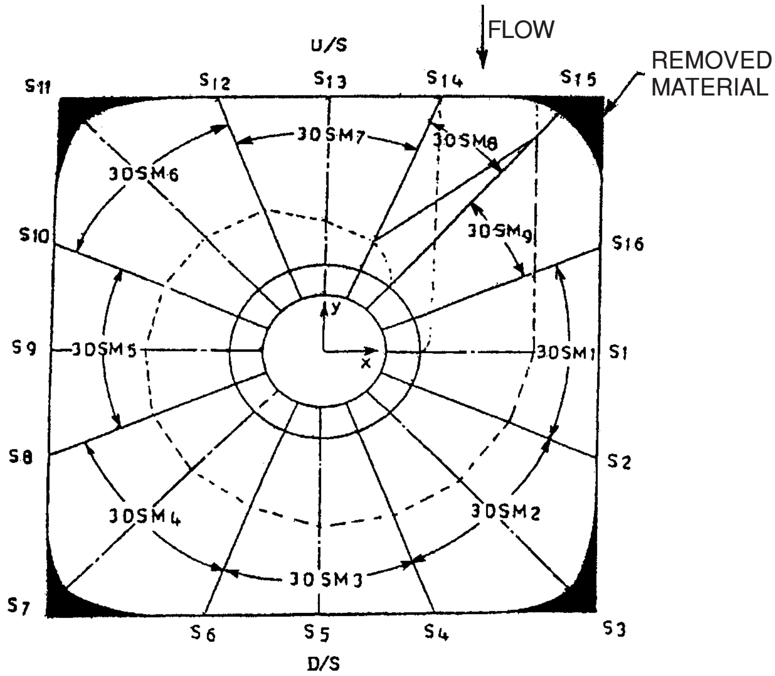


Figure 7 Nigam's model no.37 divided into sixteen slices in 2-D and nine slices in 3-D ($S_1 - S_{16}$ represent slices in 2-D and $3DS_1 - 3DS_9$ represent slices in 3-D)

3.2 Computer Programs

The Computer programs used in this investigation are as follows:

(i) TAES is an existing well tested linear elastic 3-D finite element program.
(ii) MESH3 is a 3-D automatic mesh generation program developed (in FORTRAN language) as pre-processor for TAES which discretise the structure and the model into a suitable number of elements.
(iii) PLOT3 is a graphic package developed (in FORTRAN language) to check the generated meshes by MESH3 at any desired horizontal or vertical plane.
(iv) PSPSAS is an existing linear elastic 2-D finite element program which analyses plane stress, plane strain and axisymmetric cases.
(v) MESHGEN is an existing 2-D automatic mesh generation program which is in perfect harmony (pre-processor) with PSPSAS.
(vi) COMNIX is an existing plotting program for the checking of input and graphical interpretation of the output of PSPSAS. COMNIX can plot original and/or deformed meshes, stress vectors, stress/strain contours.
(vii) BEMP is an existing boundary element program which can analyze plane strain case of linear elastic structure with specified normal and tangential tractions/displacements at the boundary.

3.3 Plane Strain and Axisymmetric Cases

The structure has been divided into sixteen equal radial slices in plan. Each vertical slice has been analysed as a separate plane strain case using eight nodded isoparametric elements. Since the thickness of each slice increases toward the outer edge of the power house structure, therefore, elements with variable thickness have been adopted in the analysis. PSPSAS computer package has been modified for provision of varying element thickness. This gives a better approximation in loads and stiffness calculations.

The simplified generator loads considered in the analyses of the structure and the model have axisymmetric nature. Therefore, slices with large cavities have also been analysed as axisymmetric cases using PSPSAS computer program.

3.4 Boundary Element Cases

Plane strain boundary element analysis of different slices have been carried out using BEMP computer program. In this study, for simplicity purposes, the concrete barrel shaft of the generator house was not included in the analysis. Each slice is considered with a constant unit thickness in the analysis [1,3].

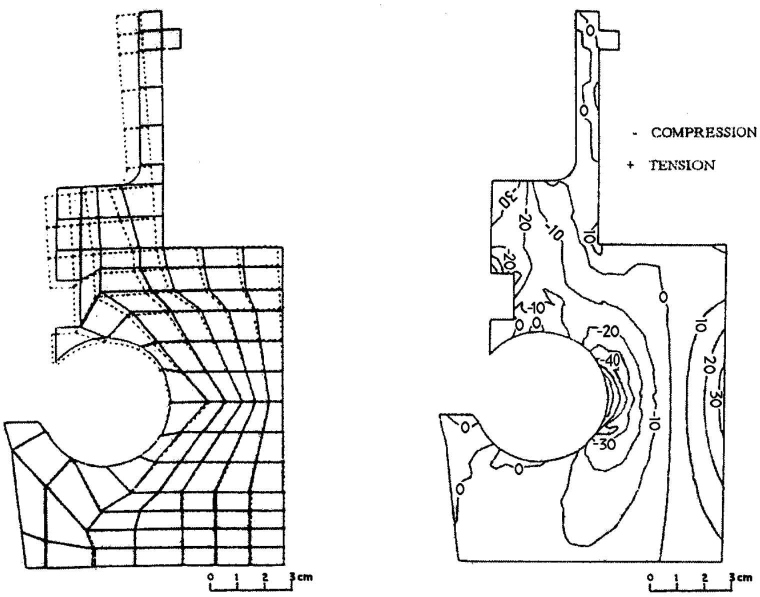
3.5 3-D Finite Element Case

Using 20 noded isoparametric brick elements, the structure has been discretised into 180 elements and with the help of TAES computer package, a full three-dimensional finite element analysis of the structure has been carried out. Similarly Nigam's model has been discretised into 81 brick elements and its analysis has been carried out.

4.0 COMPARISON AND DISCUSSION OF RESULTS

The most important results from this particular study are the stress variations. Using different numerical methods of analysis, the slices have been analysed and the stress contours at only two sections have been plotted (Figure 8). These two sections which have the maximum cavities of the scroll case in longitudinal and transverse directions, are Sec.1 (scroll case at inlet) and Sec.5 (scroll case at down stream side) respectively. Prior to this study, the relatively realistic stress modeling in the structure was the use of SCF provided by Nigam. Therefore, the stresses obtained from various analyses of this study are compared with those calculated using Nigam's SCF. One section in transverse direction (Sec. 5) and one section in longitudinal direction (Sec.1) have been selected to compare the variation of vertical stresses in the power house structure case. In the case of the model, two sections in the transverse direction and two sections in the longitudinal direction have been selected to compare the vertical stresses from various analyses.

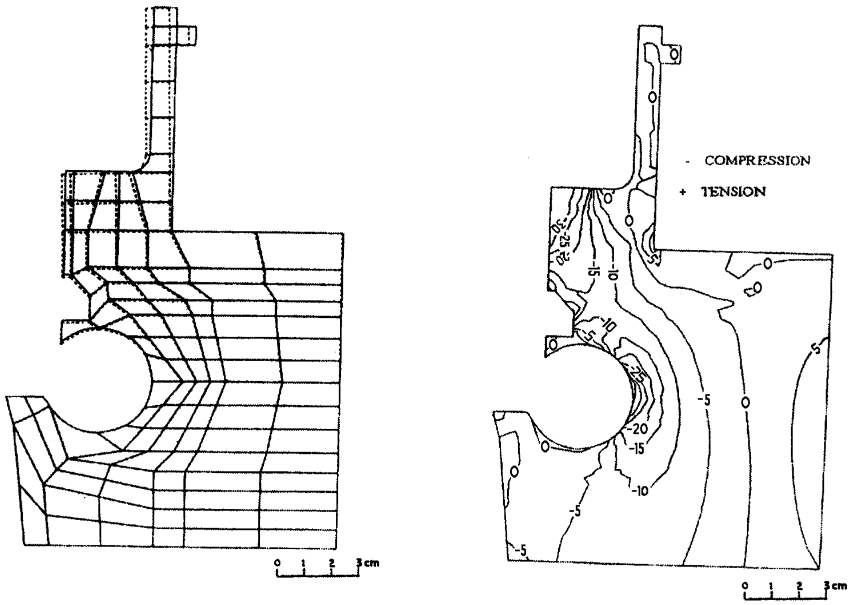
The results show that vertical stresses at scroll case side wall obtained from different analyses, both in cases of the model and the structure have almost the same pattern



Deformation 1000 Times Magnified
Scale: 1 cm = 2/3 m

Stresses in t/m^2
Scale: 1 cm = 2/3 m

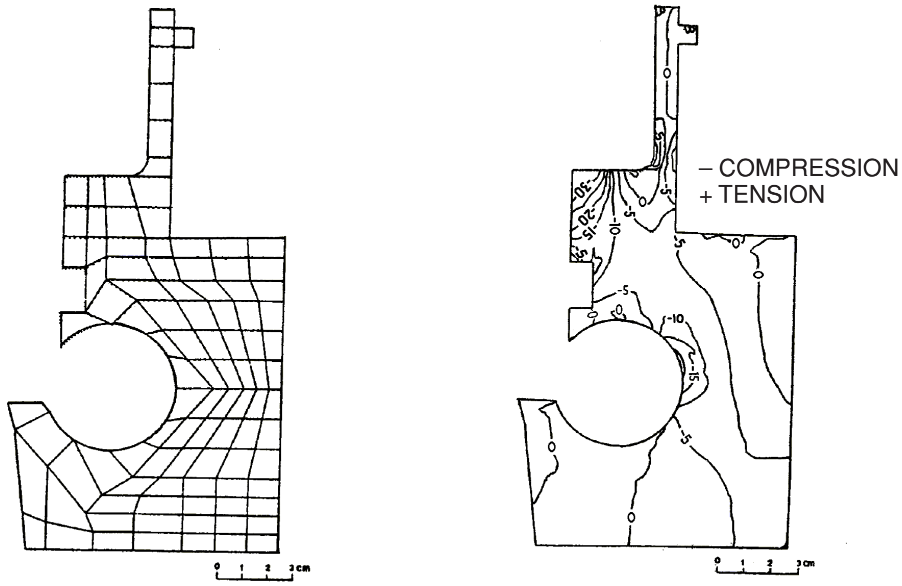
Figure 8(a) Deformed shape and contours for vertical stresses at Sec. 1, plane strain analysis



Deformation 1000 Times Magnified
Scale: 1 cm = 2/3 m

Stresses in t/m^2
Scale: 1 cm = 2/3 m

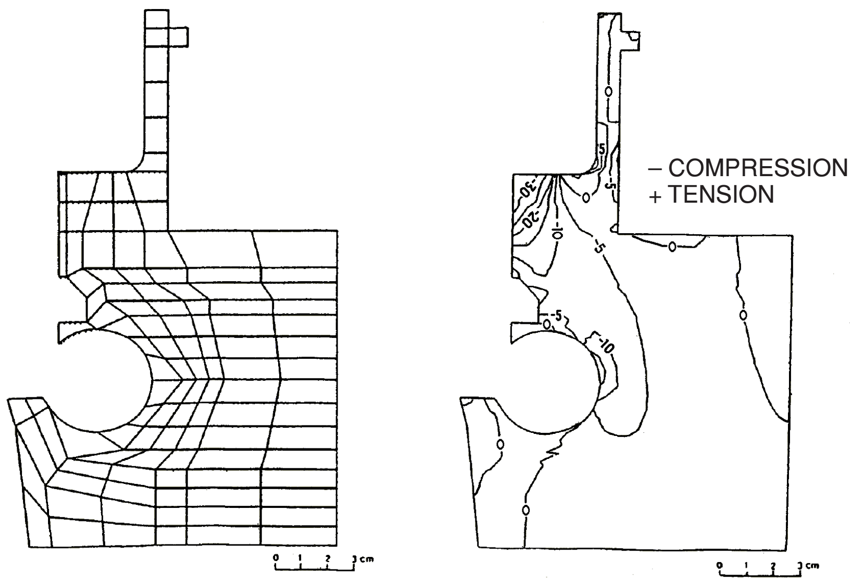
Figure 8(b) Deformed shape and contours for vertical stresses at Sec. 5, plane strain analysis



Deformation 1000 Times Magnified
Scale: 1 cm = 2/3 m

Stresses in t/m^2
Scale: 1 cm = 2/3 m

Figure 8(c) Deformed shape and contours for vertical stresses at Sec. 1, axisymmetric analysis



Deformation 1000 Times Magnified
Scale: 1 cm = 2/3 m

Stresses in t/m^2
Scale: 1 cm = 2/3 m

Figure 8(d) Deformed shape and contours for vertical stresses at Sec. 5, axisymmetric analysis

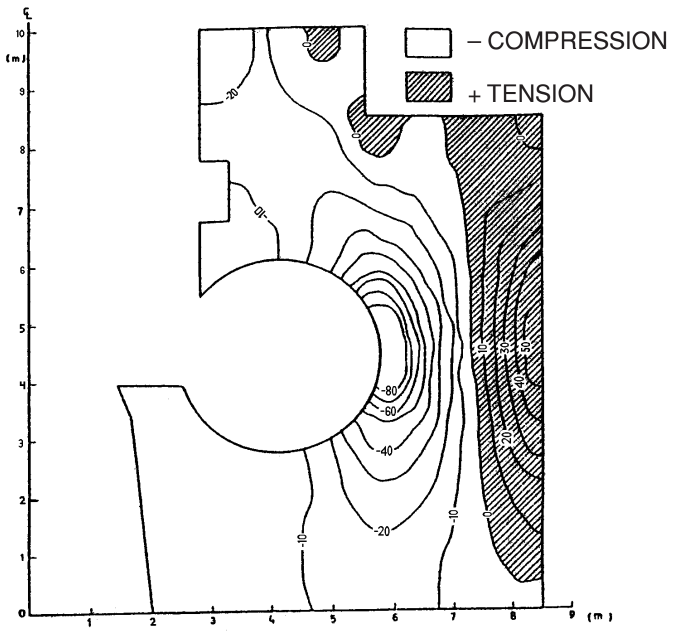


Figure 8(e) Contours for vertical stresses at t/m^2 at Sec. 1, boundary element analysis

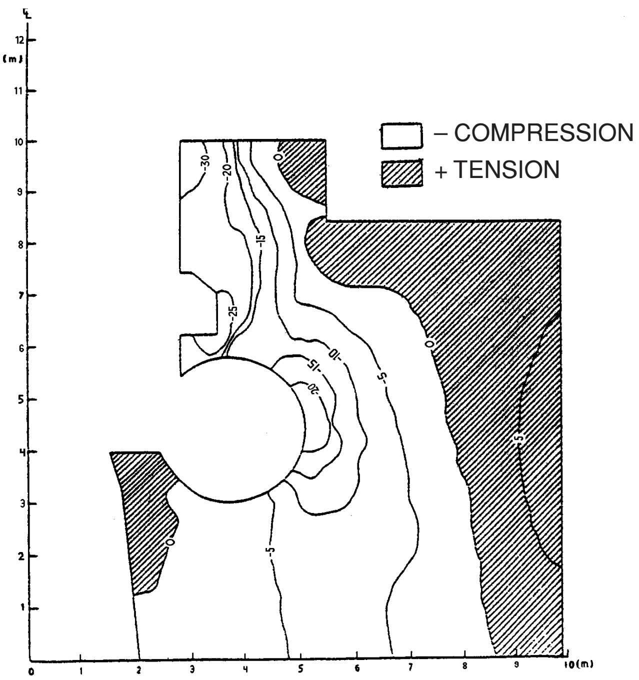


Figure 8(f) Contours for vertical stresses at t/m^2 at Sec. 5, boundary element analysis

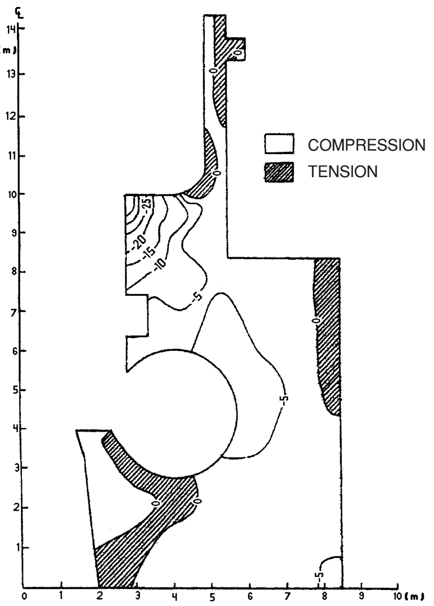


Figure 8(g) Contours for vertical stresses in t/m^2 at Sec. 1, 3-dimensional finite element analysis

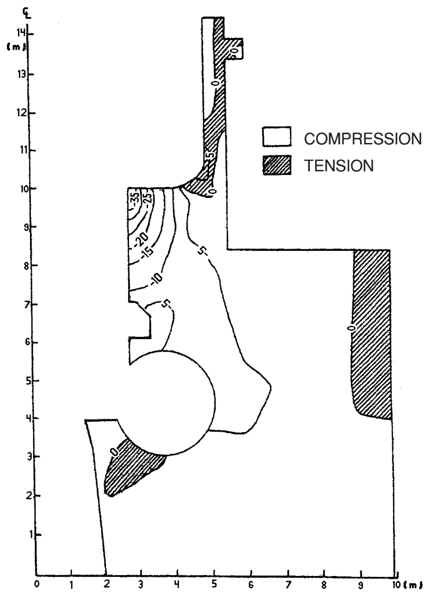


Figure 8(h) Contours for vertical stresses at t/m^2 at Sec. 5, 3-dimensional finite element analysis

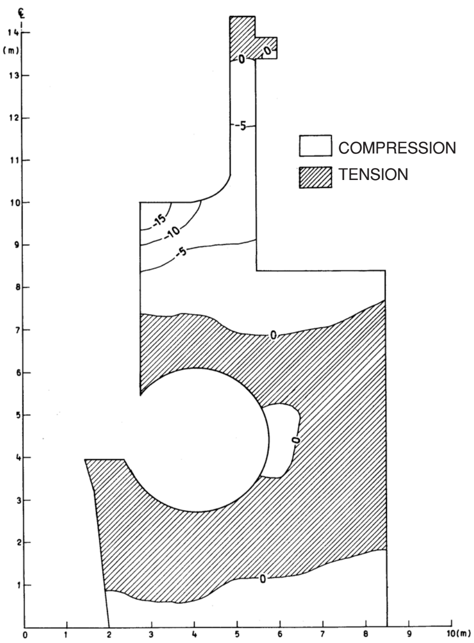


Figure 8(i) Contours for vertical stresses in t/m^2 at Sec. 1, 3-dimensional finite element analysis

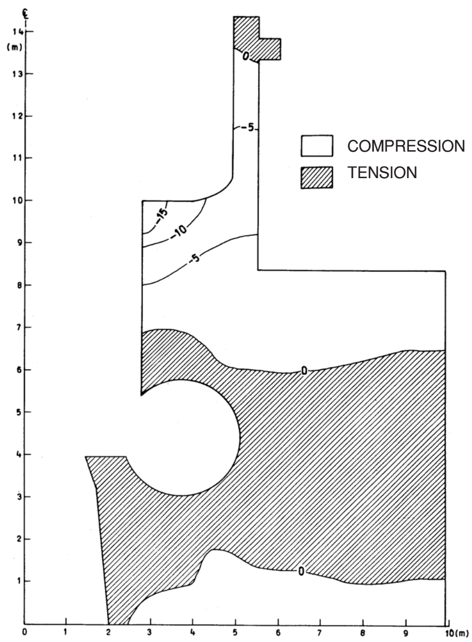


Figure 8(j) Contours for vertical stresses at t/m^2 at Sec. 5, 3-dimensional finite element analysis and nature and only their magnitudes differ, while the stresses obtained using Nigam's SCF at the same level predicts different nature of the stresses. Figure 9(a) & (b) show the graphical representation for the variation of vertical stresses from various analyses at the side wall of model.

The variation of vertical stresses at scroll case side wall from different analyses is shown in Figure 9(c) & (d).

5.0 CONCLUSIONS

The study has been carried out to give an insight into the behaviour of scroll case and generator foundation including generator room barrel shaft under the action of vertical generator loads only.

The stresses obtained using numerical methods have a vast difference with those obtained by using Nigam's SCF. The results from all cases in the present study show vertical tensile stress at outer portion of the scroll case side wall, while Nigam's SCF gives compressive stress.

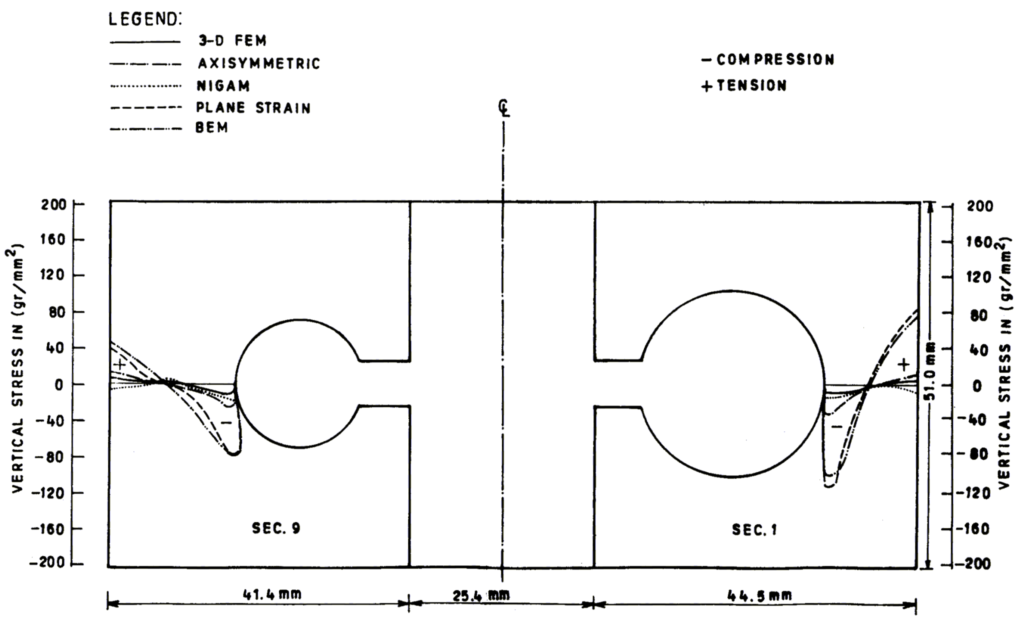


Figure 9(a) Variation of vertical stresses at longitudinal Section of Nigam's model, using different methods of analysis

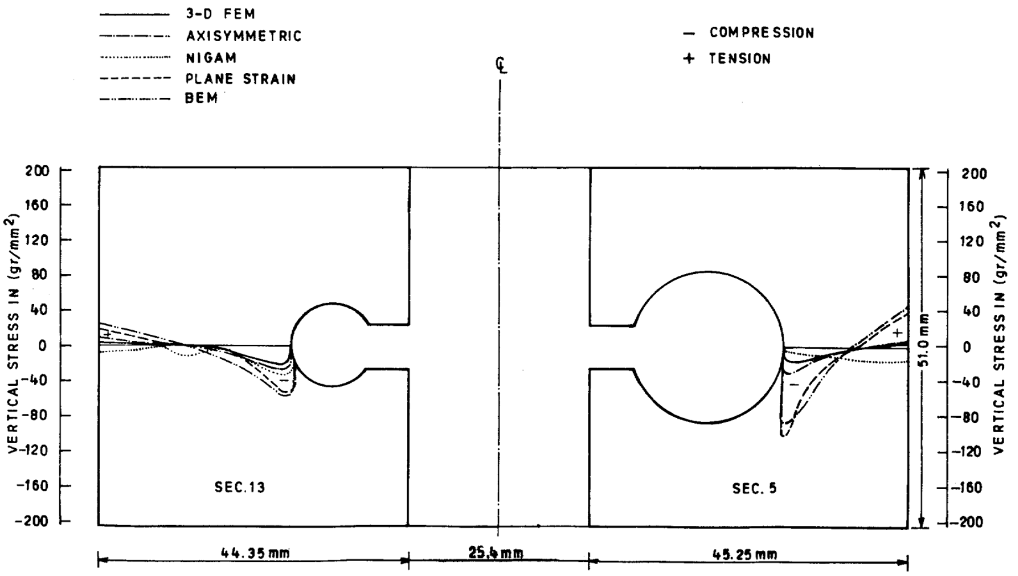


Figure 9(b) Variation of vertical stresses at transverse section of Nigam's model, using different methods of analysis

Despite being an accurate method of analysis, BEM gives higher values of stresses. The investigation shows that the vertical stresses at side wall of the scroll case obtained by BEM are 60% higher than those obtained using plane strain analysis (FEM). This may be due to the fact that in BEM, the sections have been assumed with constant unit thickness, hence BEM cannot be adopted.

3-D FEM and axisymmetric analyses of the present investigation indicate a tensile hoop stress in concrete around the scroll case and in some parts of the barrel shaft which needs reinforcement to be provided. On the other hand the conventional methods such as beam-column assumption arch and ring arrangement cannot predict such stresses therefore, it will not be appropriate to adopt them in the design of the structure.

Axisymmetric analysis gives results much closer to the 3-D FEM results. Therefore, considering its ease and simplicity, it is an acceptable replacement for the cumbersome and tedious 3-D FEM analysis of the structure.

This study pinpoints the stress concentration especially tensile stress zones in concrete around scroll case and generator foundation which require to be reinforced. However, for the final design of the structure all of the applied loads and their different combinations have to be considered.

Being simple and easy to handle, plane strain and axisymmetric analyses using slicing technique are recommended for the analysis of the structures. These methods can be used to determine the stress concentration zones and their maximum values considering different load combinations in all types of the surface hydro-power stations.

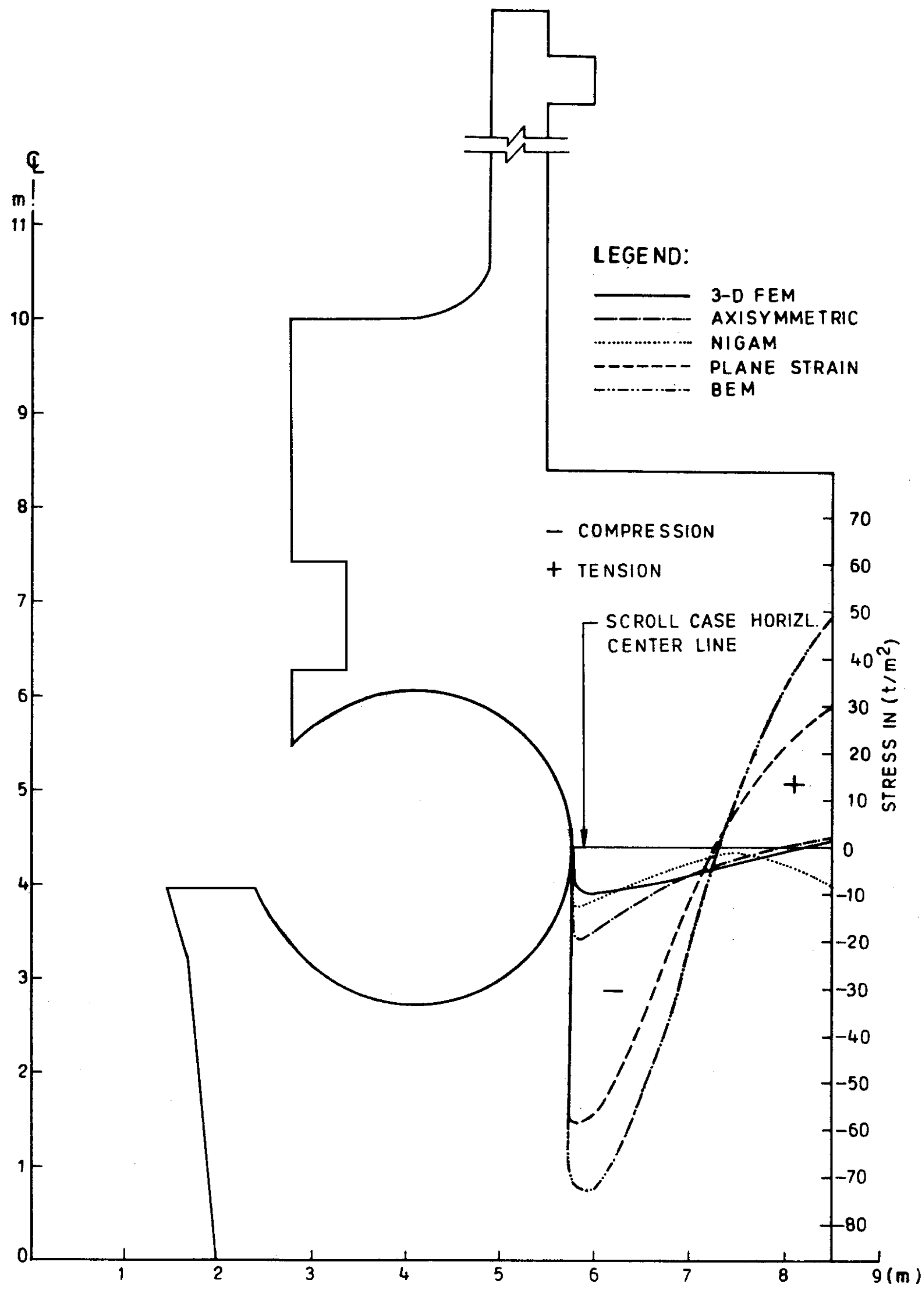


Figure 9(c) Variation of vertical stresses in Sec. 1 of the power house at scroll case level, using different methods of analysis

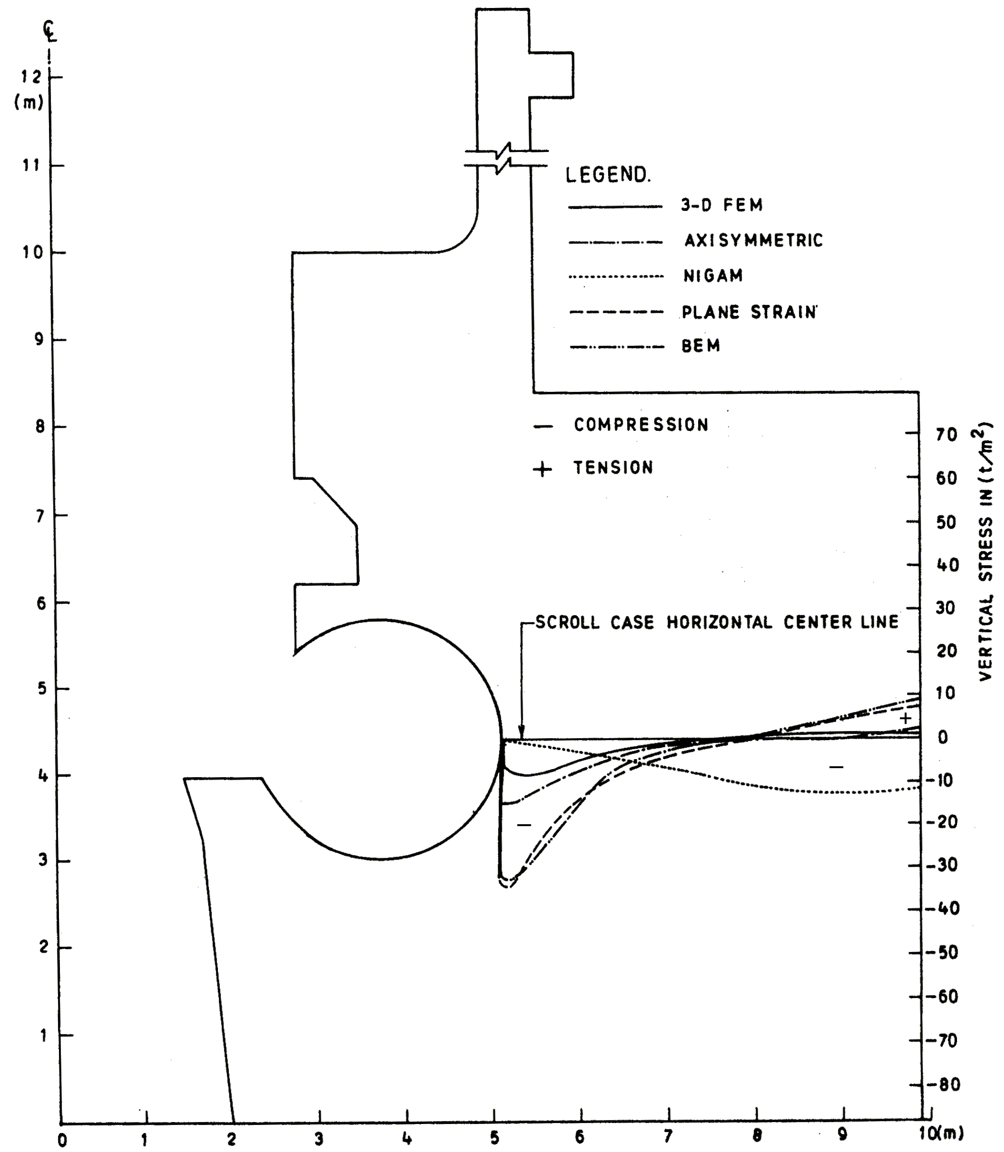


Figure 9(c) Variation of vertical stresses in Sec. 1 of the power house at scroll case level, using different methods of analysis

REFERENCES

[1] Bakhteri, J. 1992. *"Analysis of Generator Foundation of a Hydro-Electric Power House"*. Ph.D. Thesis, University of Roorkee, India.

[2] Bakhteri, J., G. C. Nayak, and P. N. Godbole. 1994. "Finite Element Analysis of Scroll Casing of a Hydro-Electric Power House". Proceeding of the Third Regional Conference on Computer Application in Civil Engineering, Kuala Lumpur, Malaysia.

[3] Crouch, S. L., and A. M. Starfield. 1983. *Boundary Element Method in Solid Mechanics*. London: George Allen & Unwin.

[4] Ilyinykh, I. I. 1985. *Hydroelectric Stations.* Moscow: Mir Publications.

[5] Nigam, P. S. 1979. *Handbook of Hydro-Electric Engineering, Nem Chand and Brothers.* India: Roorkee.

[6] Nigam, P. S. 1972. *"Stress Distribution in Concrete Around Spiral Case and Draft Tube of a Hydel Power Station"*. Ph.D. Thesis, University of Roorkee, India.

[7] Zienkiewicz, O. C. 1972. *The Finite Element Method.* New York: McGraw Hill.

[8] IS: 4247 Part I, Part II & Part III. 1978. *Code of Practice for Structural Design of Surface Hydel Power Stations*. New Delhi: Indian Standard Institution.

[9] IS: 7418, 1974. *Criteria for Design of Spiral Casing (Concrete and Steel)* New Delhi: Indian Standard Institution.

[10] IS: 7207, 1974. *Criteria for Design of Generator Foundation for Hydel Power Stations*. New Delhi: Indian Standard Institution.