Zeitschrift: IABSE reports = Rapports AIPC = IVBH Berichte
Band: 54 (1987)

Artikel: Analysis of underwater tunnel for internal gas explosion
Autor: Meyer, Christian
DOI: https://doi.org/10.5169/seals-41954

Analysis of Underwater Tunnel for Internal Gas Explosion

Calcul d'un tunnel sous-marin dans le cas hypothétique d'une explosion interne de gaz

Berechnung eines durch Gasexplosion belasteten Unterwassertunnels

Christian MEYER
Assoc. Prof. of Civil Eng.
Columbia University
New York, N.Y., U.S.A.

Christian Meyer received his M.S. and Ph.D. degree from the University of California at Berkeley and spent 8 years in the building and nuclear power industry before joining Columbia University. His research interests are in computer analysis of structures, earthquake engineering and analysis and design of concrete structures.

SUMMARY

Nonlinear finite element analysis of concrete structures is a new tool available for practitioners in situations where conventional simple methods of analysis may not be adequate. This paper illustrates the use of DIANA, a comprehensive state-of-the-art finite element program system, for the nonlinear dynamic analysis of an underwater tunnel subjected to a hypothetical internal gas explosion. Emphasis is placed on the care with which the engineer has to verify the correctness of the program, the model and the analysis results.

RÉSUMÉ

Le calcul statique de structures en béton à l'aide des méthodes aux éléments finis non linéaires est un nouveau moyen à disposition des praticiens dans les situations où les méthodes simples conventionnelles de calcul statique peuvent ne pas être satisfaisantes. La contribution illustre l'application de DIANA, un système global de programmes aux élémets finis pour le calcul dynamique non-linéaire d'un tunnel sous-marin sous la charge hypothétique d'une explosion de gaz à l'intérieur du tunnel. L'auteur souligne l'importance des précautions à prendre par l'ingénieur, afin de vérifier l'exactitude du programme, du modèle et des résultats d'analyse.

ZUSAMMENFASSUNG

Nichtlineare Finite Element Berechnungen von Betonkonstruktionen bieten dem Praktiker neue Möglichkeiten, wo die herkömmlichen vereinfachten Berechnungsmethoden nicht ausreichend sind. Dieser Aufsatz dient als Illustration einer Anwendung von DIANA, einem umfassenden Finite Element Programmsystem auf dem letzten Stand der Forschung. Als Beispiel dient die nichtlineare dynamische Berechnung eines durch Gasexplosion belasteten Unterwassertunnels. Besonders wird auf die Sorgfalt hingewiesen, mit welcher der Benutzer das Programm, das Rechenmodell und die Berechnungsergebnisse verifizieren muss.

1. INTRODUCTION

Designing reinforced concrete structures implies the capability to analyze such structures not only for the loads they are expected to carry during their design life, but also to determine the factor of safety against failure under overload. For most engineered structures, designers can rely on simple and proven analysis methods to do that. Behavior under service load is analyzed as a rule by linear elastic methods. These are routinely extrapolated for the determination of ultimate load capacities, whereby the structure's redistribution of loads is often accounted for in a more or less intuitive way. Designers are periodically confronted with unusual structures, for which their past experience is not sufficient for satisfactory treatment. Structures of this kind are generally surface structures such as plates, shells, deep beams, and often massive or thick-walled structures. To evaluate the performance of such structures, designers have the choice of further relying on their intuition, borne out of experience, or they may elect a novel analytical approach: nonlinear finite element analysis. Until only a few years ago they did not have this choice. Neither were the computational tools adequate for such tasks, nor were the properties known sufficiently well to allow the development of realistic mathematical models.

On both counts considerable progress has been made in recent years [1,2]. Nonlinear finite element solution techniques have matured to the extent that it is now possible to compute the highly nonlinear response of systems with thousands of degrees of freedom, effectively placing a new analysis tool at the disposal of engineering practitioners. Developments in hardware technology have made it possible to carry out such computations on VAX-size supermini computers. Yet, for all this progress, numerous pitfalls call for considerable caution. First, the advances in nonlinear computational mechanics were such that the realistic modeling of material properties has become the primary limitation of this analytical approach. Second, the finite element idealization of structures for nonlinear analysis requires considerable skills on the part of the analyst. Third, the vast amount of numerical computations involved are still taxing computer resources to such an extent, that full nonlinear finite element analyses will be restricted to unusual and special structures for some time to come.

In view of these concerns it is important that this new technology be introduced cautiously, lest it receive adverse publicitiy before having had a fair chance to prove itself in engineering practice. In the Netherland, two related developments are of interest in this regard. First, there is the continuing development of program DIANA [3], which contains state-of-the-art material models for reinforced concrete and efficient numerical solution algorithms. Second, the Netherlands Committee for Research, Codes and Standards for Concrete (CUR-VB) is funding an effort to demonstrate the capabilities of DIANA with an "Example-Book" [4], a publication containing a number of realistic examples taken from engineering practice. This author had the opportunity to participate in this effort while on Sabbatical leave in Delft, by analyzing an underwater tunnel for an internal gas explosion.

It is the purpose of this paper to use this example as an illustration both of the potential and the difficulties of nonlinear finite element analysis of concrete structures. The material models built into programs like DIANA are discussed briefly, followed by some comments on practical nonlinear finite element analysis in general. The analysis example itself is presented here primarily as an illustration of how one should approach a problem of this kind in a practical setting.

2. MATHEMATICAL MODELS FOR REINFORCED CONCRETE

The first essential prerequisite for any realistic analysis is a thorough understanding of reinforced concrete behavior under load. This includes the following specific aspects:

1. plain concrete behavior under arbitrary stress states, i.e., constitutive behavior, cracking and crack propagation, and possible failure mechanisms in compression;
2. reinforcing steel behavior;
3. bond behavior at the steel-concrete interface;
4. shear transfer mechanisms across cracks;
5. long-term deformations due to creep and shrinkage;
6. dynamic strain rate effects in the case of impact and blast loads;
7. strength and stiffness degradation effects accompanying large inelastic cyclic loads.

New experimental techniques have made available a wealth of data that has improved our understanding of concrete behavior and is suitable to support the development of improved material models. These are based on a variety of different theories, such as nonlinear elasticity, plasticity, viscoplasticity, plastic-fracturing, and endochronic theories. References [1,2] give a broad overview of these theories and some of the models that have been used successfully in recent years to analyze realistic reinforced concrete structures. However, a word of caution is in order, because many of these models are still undergoing development and therefore should be used only with great care. It is appropriate to refer in this context to the international competition [5] which demonstrated that wide scatter of results is not limited to experimental investigations of concrete structures, but applies to analytical studies as well. For this reason it is inappropriate to place unrealistic expectations in the degree of accuracy with which these new models and theories can simulate concrete behavior as observed in the laboratory.

3. NONLINEAR FINITE ELEMENT ANALYSIS

The finite element method has invaded engineering practice in a relatively short time. Most engineering offices in the United States now have access to major finite element program systems [6]. Finite element analyses in engineering practice are generally limited to linear elastic response. Even then, the development of proper models requires a considerable amount of skill and experience on the part of the analyst. Possibly the most difficult part of this task is an independent verification of the correctness of the analysis results. All too often, analysts, for a number of reasons, fail to undertake this important step and therefore are bound to accept and use output results, even if these are completely wrong. In an effort to help analysts in proper techniques of finite element idealization and output verification, a comprehensive guide for linear static and dynamic analysis has just been published [7].

In the case of nonlinear finite element analysis the difficulties are multiplied for a number of reasons. First, the material models are incomparably more complicated, requiring of the analyst an intimate knowledge of the material being modeled and of the details of the material model itself. In many cases the program user is offered a number of alternate material models, and he has to have the proper training and experience in order to make the right choices. Second, the powerful numerical angorithms now available in many nonlinear analysis programs, generally lack the "ruggedness" of linear analysis algorithms, i.e., they are much more vulnerable to improper use. As a rule, the user has to be intimately familiar with the algorithm's idiosyncrasies, its limitations and range of applicability. In linear analysis this is much less the case. A third complication is the greatly increased difficulty of interpreting and verifying the output results. For these reasons it is essential that the analyst, even if highly trained and expert in his field, proceeds very cautiously to verify step by step the correctness of the program, its algorithms and material models, then the finite element model of the structure to be analyzed, and finally the analysis results themselves. The need for this painstaking verification process is the main source of the high cost of nonlinear analysis of realistic structures, which limits the application of this sophisticated analysis tool to very unusual, important, or expensive structures. The example presented below shall serve as an illustration of what the author believes is a methodical procedure to solve a difficult problem.

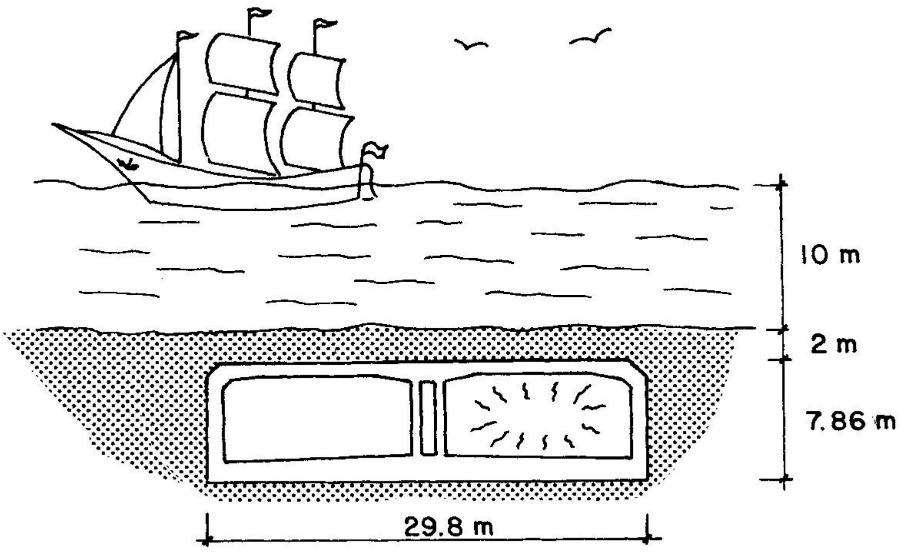
4. DYNAMIC ANALYSIS OF UNDERWATER TUNNEL

4.1 General

Road tunnels that pass under waterways are very common in the Netherlands. They are normally designed to resist the loads associated with soil and water pressure. In the event of an internal gas explosion, the tunnel experiences a load reversal for which it may not be adequately reinforced. Thus, the question whether hazardous cargo should be permitted to pass through such tunnels is of some importance. The Dutch Public Works Department (Rijkswaterstaat) has developed standard tunnel cross sections that are widely used throughout the Netherlands. Figure 1 shows a typical cross section and material properties. The strength parameters listed in Fig. 1b are based on experimentally determined values and a 20% allowance for the strain-rate effect.

It was the objective of this analysis to predict the response of the tunnel to the pressure load associated with a hypothetical internal gas explosion. The solution of a problem of this nature requires a careful step-by-step approach, with continuous verification of the correctness of the program, the finite element model, and the results obtained. In order to achieve these goals, the following analyses were performed:

1. a linear elastic static frame analysis of the entire tunnel cross section;
2. a linear static finite element analysis of a segment of the tunnel roof;



a) Typical Tunnel Section

Concrete

$E_c = 22{,}000\ \mathrm{N/mm^2}$

$\nu = 0.2$

$f_t' = 3.36\ \mathrm{N/mm^2}$

$f_c' = 30\ \mathrm{N/mm^2}$

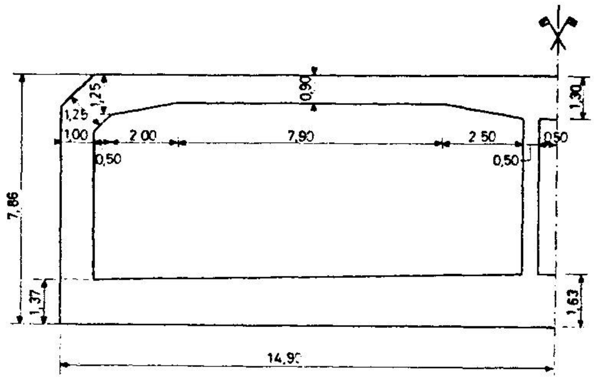
$\varepsilon_{tu} = 0.001$

Reinforcing Steel

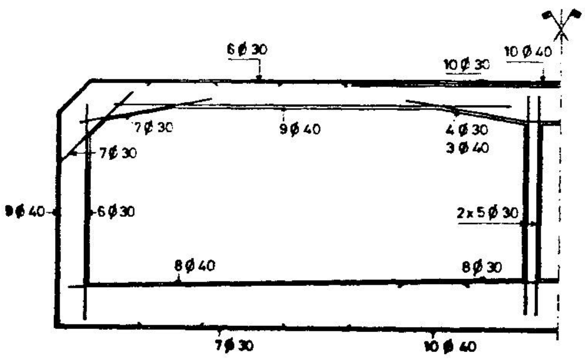
$E_s = 210{,}000\ \mathrm{N/mm^2}$

$f_y = 528\ \mathrm{N/mm^2}$

b) Material Properties

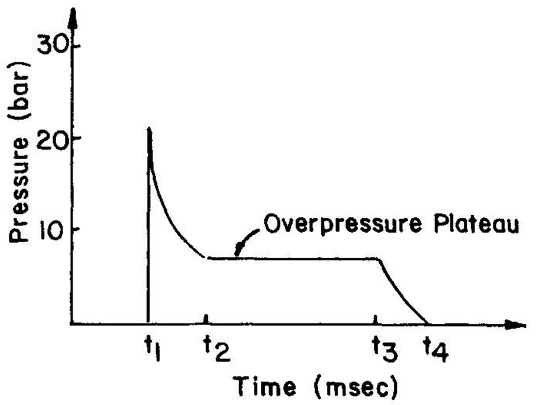


c) Dimensions

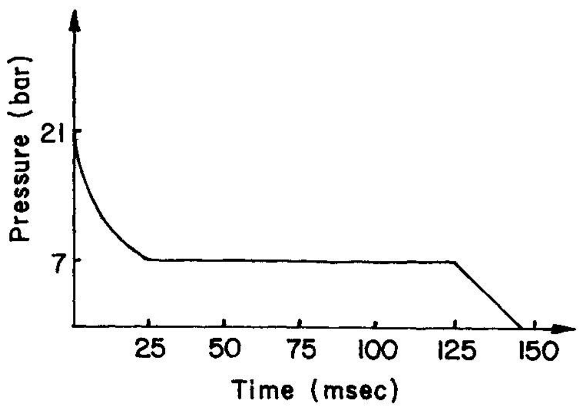


d) Reinforcement for 1.5m Wide Section

Fig.1 Typical Tunnel Section



a) Generic Pressure Time History



b) PressureTime History Used In This Analysis

Fig.2 Assumed Pressure Time History

3. a nonlinear static finite element analysis of the same tunnel roof segment;
4. an eigenvalue analysis of the finite element model;
5. a nonlinear dynamic time history analysis of a grossly simplified finite element model;
6. the final nonlinear dynamic time history analysis of the actual finite element model.

At each step measures were taken to verify that the analysis results were reasonable. For this purpose it was very helpful that a 1:5 scale model of a particular tunnel section had been tested at the TNO-IBBC Institute in 1976 [8]. The documentation of this work contains detailed information on dimensions, reinforcement, material properties and service loads on the prototype structure [8].

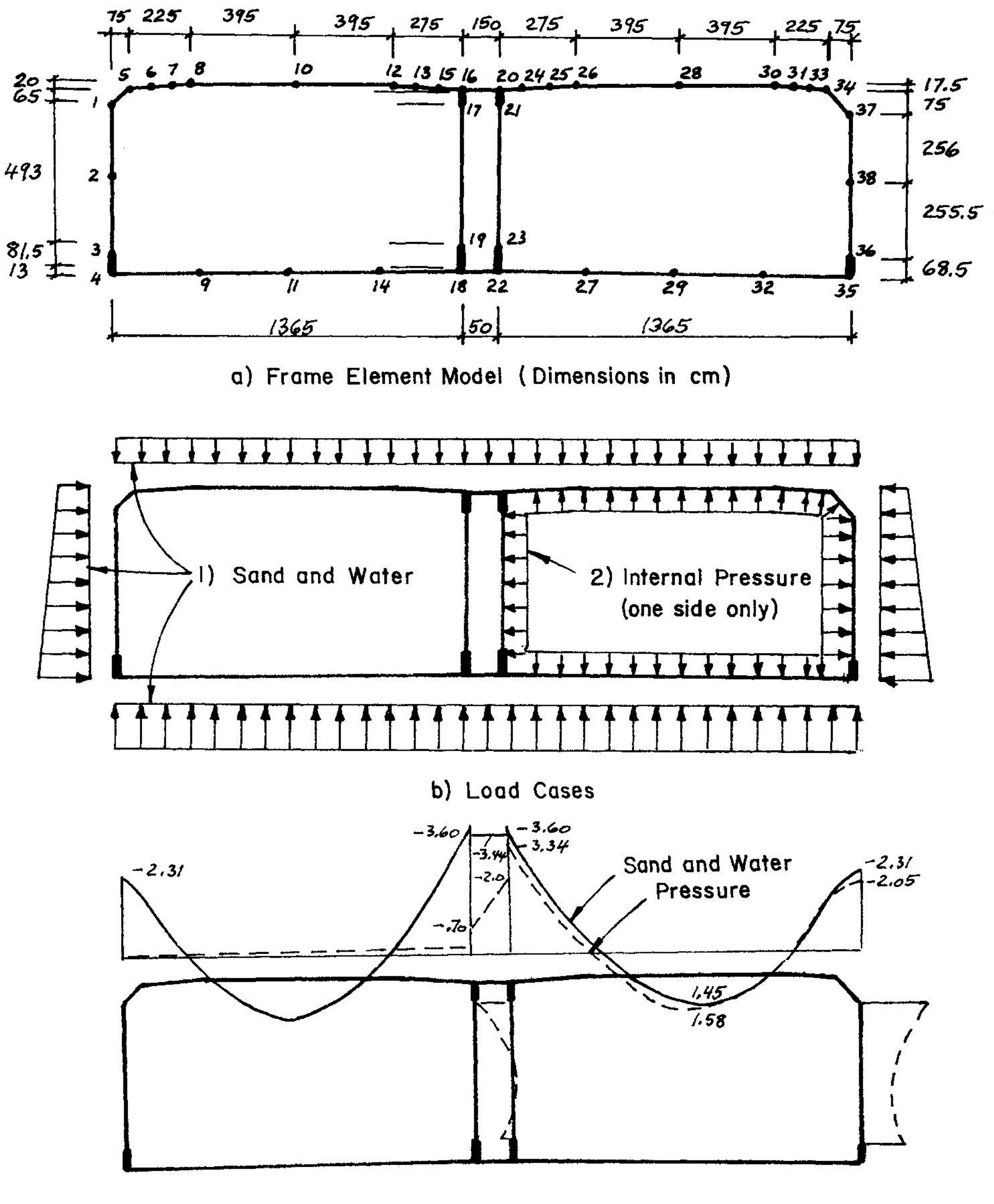
4.2 Loading

Little is known about the dynamic pressure loads generated by internal gas explosions. In a joint Dutch/Belgian effort, a series of tests have been conducted on an experimental tunnel of 1.8 m by 1.8 m cross section and 27 m length [9]. From these experiments it was possible to identify the following characteristics of a pressure time history; Fig. 2a,

1. The shock front is for all practical purposes vertical, i.e., the pressure increases instantaneously from ambient to a peak value of about 25 bar.
2. The peak pressure drops rapidly to an overpressure plateau, following approximately a parabolic shape.
3. The overpressure remains approximately constant at the value of 6 to 7 bar. This value can be computed from the gas-air mixture, considering the energy released during the chemical reaction. The length of the plateau is a function of the time needed to vent the overpressure.
4. Once the depressurization of the tunnel starts, the decrease of overpressure follows again an approximately parabolic shape.

The scaling of these experimental pressures for tunnels of different dimensions is not straightforward. Concerning the tunnel cross-sectional dimensions it can be argued that the energy released per unit volume is invariant, therefore both the peak pressure and the plateau pressure are approximately independent of the cross-sectional area, assuming the entire cross-section is filled with combustible gas. In contrast, the time of depressurization onset should be an approximately linear function of tunnel length, because the travel times of both the shock wave and its reflection are functions of tunnel length, again assuming the entire tunnel is filled with gas. Assuming further that detonation commences at the center of the 320 m long tunnel, and that the tunnel section to be analyzed is situated at the quarter point, i.e. 80 m from the tunnel exit, the pressure time history of Fig. 2b was arrived at. The shock wave velocity is about 2000 m/sec, and the velocity of the depressurization wave is about half that much, because depressurization is associated with fluid flow, a considerably slower process.

It is noteworthy that both the peak pressure of 25 bar (2.5 N/mm^2 or 362 psi) and the plateau pressure of 7 bar (0.7 N/mm^2 or 101 psi) applied for the duration of 0.1 sec represent a formidable



a) Frame Element Model (Dimensions in cm)

b) Load Cases

c) Roof Bending Moments

Fig.3 Linear Elastic Static Analysis of Frame Element Model load which a conventionally reinforced structure is unlikely to survive without severe damage.

4.3 Modeling of Structure and Model Verification

The first analysis step was a linear static analysis of a simple frame model of the entire cross section, Fig. 3, for, 1) soil and water pressure and, 2) internal pressure. Because of the similarity of the moment distributions for the two load cases it could be justified to model only a quarter of the entire roof slab for the finite element analysis and to apply boundary conditions valid for both load cases.

In the finite element model, Fig. 4, 45 eight-noded plane stress elements CQ16M (in the final analysis, plane strain elements CQ16E) were used for the concrete, and the reinforcement was modeled by 34 bar elements as shown, resulting in a total of 172 nodes with 344 potential degrees of freedom. In order to compare the analysis results for the frame element and finite element models, it was necessary to account for the following modeling differences.

1. The right face of the finite element model was fixed against rotation, while the flexibility of the outside walls in the frame model shifted the point of maximum positive moment and zero rotation to the right. To correct the finite element model, the midspan rotation of the frame model was input as a specified rotation of the right face of the finite element model.
2. The axial deformations of the vertical walls were included in the frame element model but not in the finite element model.
3. The effect of steel reinforcement on the roof stiffness was explicitly accounted for in the finite element model, while in the frame element model only gross moments of inertia were used.
4. The representation of the haunched segment of the tunnel roof by a series of prismatic beams introduces a considerable error, which can be reduced by increasing the number of beam elements.

Once all of these factors were taken into consideration, both moments and displacements obtained by the two models agreed to within 15%.

As step three a nonlinear static analysis of the finite element model was performed, because experimental data were available for comparison and further model verification.

In the experiment, the applied loading simulated the service load distribution of Fig. 3b (load case 1) and was increased proportionally in stages, in multiples of the actual service load level. At each stage, the load was held constant for about 40 min to permit creep deformations to take place. Thereafter, the deformations were held constant for another 80 minutes for the taking of measurements. After this, the load was reduced by about 90% and increased again to the previous displacement level. 10,000 load cycles were thus applied, and the whole precedure repeated for the next load level. The five levels of 1.0, 1.2, 1.4, 1.6 and 1.7 service loads are illustrated in Fig. 5. Failure was initiated by large diagonal shear cracks and ended by crushing of concrete in the highly stressed corner where the roof joins the vertical wall.

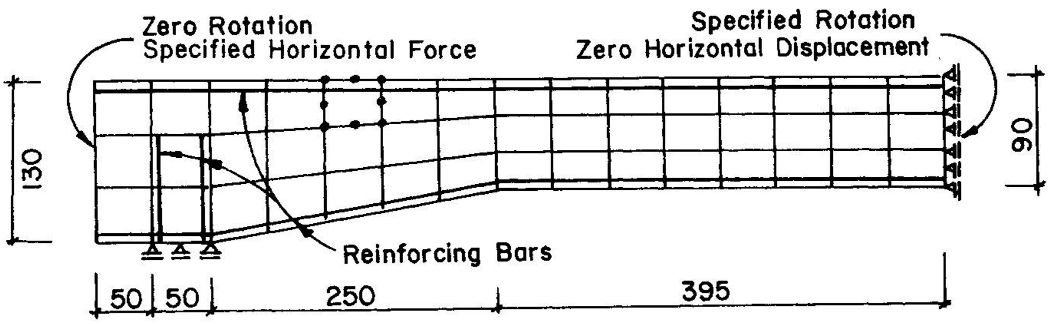


Fig.4 Finite Element Model of Tunnel Roof

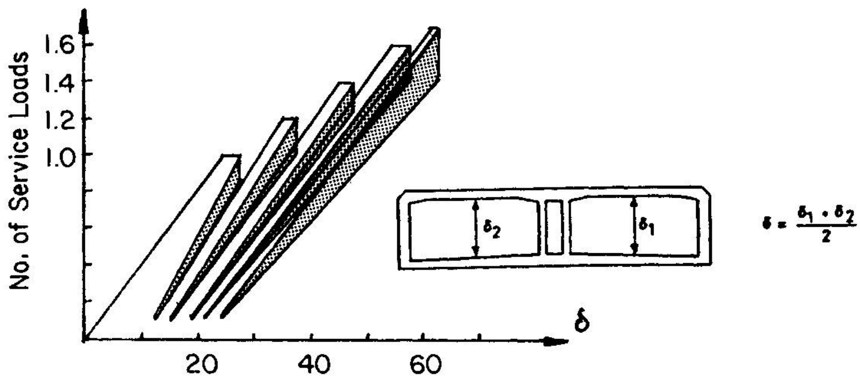
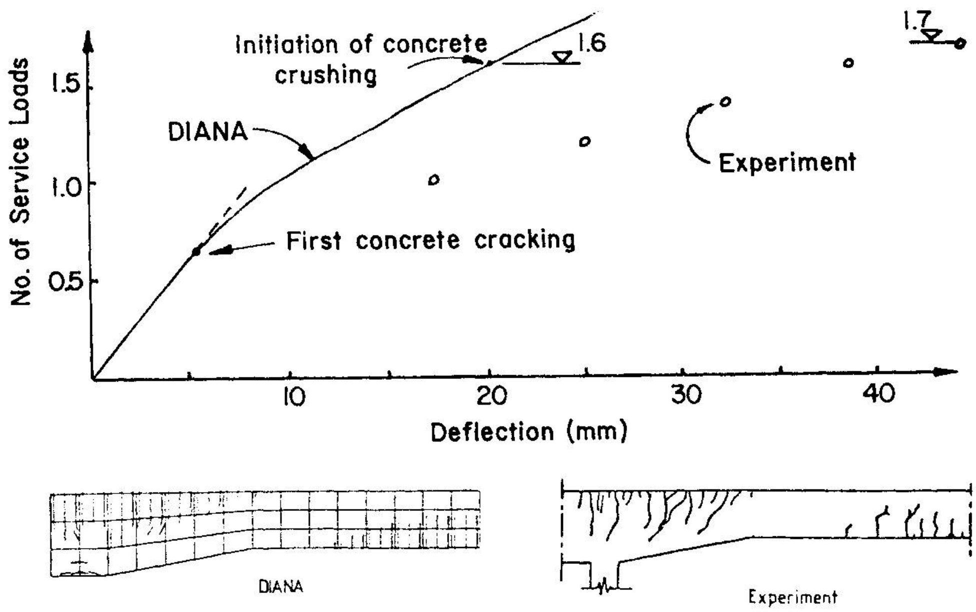


Fig.5 Load History for Scale Model Experiment



Cracking at 1.7 Service Loads

Fig.6 Experimental and Analytical Load-Deflection Information

The computed deflections did not agree particularly well with the experimental values, Fig. 6. But when comparing these results, the following factors have to be taken into consideration.

1. Experimental results were incompletely documented (see Fig. 5).
2. The large number of load cycles in the experiment resulted in cumulative damage which could not be simulated by the analysis for monotonic loading.
3. The analysis did not attempt to reproduce the creep deformations which took place in the experiment.
4. Concrete cracking can be expected to cause some moment redistribution and thus affect the boundary conditions for the finite element model, which were held constant throughout the analysis.
5. From the documentation of the experiment it was difficult to determine to what degree of accuracy all laws of similitude have been satisfied.

Even though the analysis tended to overestimate the stiffness of the structure, cracking patterns were reproduced rather accurately, and also the failure mode and failure load level agreed remarkably close, Fig. 6. It was primarily this encouraging agreement which gave rise to the confidence that it was possible to use DIANA to compute the tunnel response up to failure.

An eigenvalue analysis of the finite element model furnished mode shapes and frequencies which were in good agreement with an approximate beam solution, Fig. 7. For this and the subsequent analyses, the mass of the 2 m soil and 10 m of water was concentrated as lumped masses at the nodes along the upper boundary of the model. For the response of the structure to the primary shock load this approximation was felt to be permissible, and an involved fluid-structure interaction analysis was not justified.

The last preliminary analysis was a complete time history analysis of the grossly simplified finite element model of Fig. 8, which was very useful for familiarization with the program's dynamic analysis options and numerical algorithms, and for a first estimate of the structure's dynamic response. This analysis completed the confidence building preparation for the final analysis.

4.4 Final Nonlinear Dynamic Analysis

The final analysis consisted of 150 time steps of $\Delta t = 1.25$ msec. The adequacy of this choice of time step size was verified in a second run with 100 time steps of $\Delta t = .625$ msec which led to almost identical response results. The acceleration, velocity and displacement time histories of the roof midspan section are plotted in Fig. 9. These and the other output results permitted the following observations.

1. The first impact experienced by the structure was the axial load applied at the left boundary which is a result of the pressure on the vertical walls. This tensile impact wave propagated to the right at about 737 m/sec, causing large-scale concrete cracking in its wake and reaching the right boundary after only 7 time steps, long before the roof had any time to respond in bending to the upward pressure, Fig. 10.
2. The "concrete cracking wave" was followed by a somewhat slower

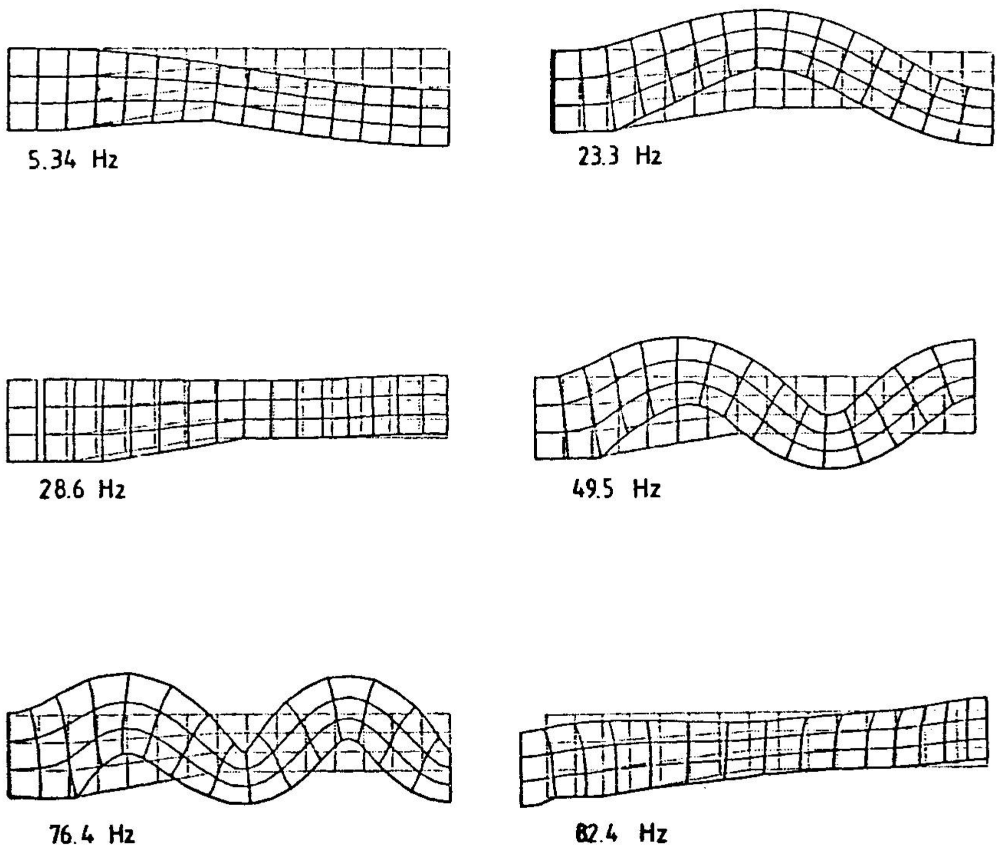


Fig.7 First Six Modes and Frequencies of Finite Element Model

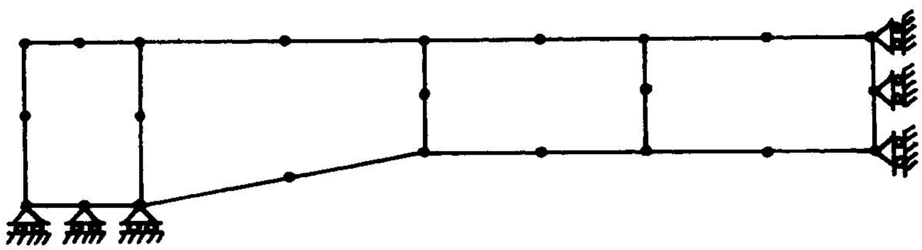


Fig.8 Grossly Simplified Finite Element Model

"steel yield wave," which caused the first steel bar to yield in the fourth time step and reached the midspan section after 22 time steps.

3. The steel stresses in the two vertical reinforcing bars, which tie the roof slab into the vertical walls, are plotted as functions of time in Fig. 11. Initially these two bars provide a fixed end moment, but as the vertical pressure tends to lift the roof off its supports, also the tensile stress in the left bar builds up.
4. Concrete stresses were not critical at any time of the analysis. The combination of flexure with axial tension forced the reinforcing steel to resist most of the load.
5. The results tend to point to the conclusion that the tunnel roof is not likely to survive a gas explosion of the kind stipulated in Fig. 2. The weakest detail appears to be the amount of vertical reinforcement which cannot prevent the vertical pressure from lifting the roof off its supports. Also, the large rotations in the plastic hinges above the support and at midspan, are associated with midspan deflections as large as 28.5 cm after 150 time steps (0.1875 sec), which can only be interpreted as failure.

5. CONCLUSIONS

Nonlinear finite element analysis of reinforced concrete structures is a new rational tool of analysis in situations where the more simplified methods are difficult or impossible to apply. In this paper, the potential of this tool has been illustrated by applying it to the dynamic analysis of an underwater tunnel subjected to an internal gas explosion. Emphasis was placed on the care with which the finite element model has to be verified and the analysis results checked for consistency and reasonableness. The time and effort required for such analyses are typically justified only for very unusual structures or situations.

Concerning the particular structure analyzed herein it was shown that both the failure mode and failure load level for service-type loads as recorded in a scale experiment were reprodued quite well. Moreover, it was possible to simulate failure under a highly dynamic blast load of a structure that was not designed for this kind of loading.

6. ACKNOWLEDGMENT

The analysis reported herein was performed during the author's Sabbatical stay at the TNO-IBBC Institute in Delft, The Netherlands. The author is indebted for the support of Prof. J. Blaauwendraad of the Delft Technical University and the DIANA development team, particularly G.M.A. Kusters and J.G. Rots.

REFERENCES

1. "Finite Element Analysis of Reinforced Concrete," Task Committee on Finite Element Analysis of Reinforced Concrete Structures, ASCE, New York, 1982.

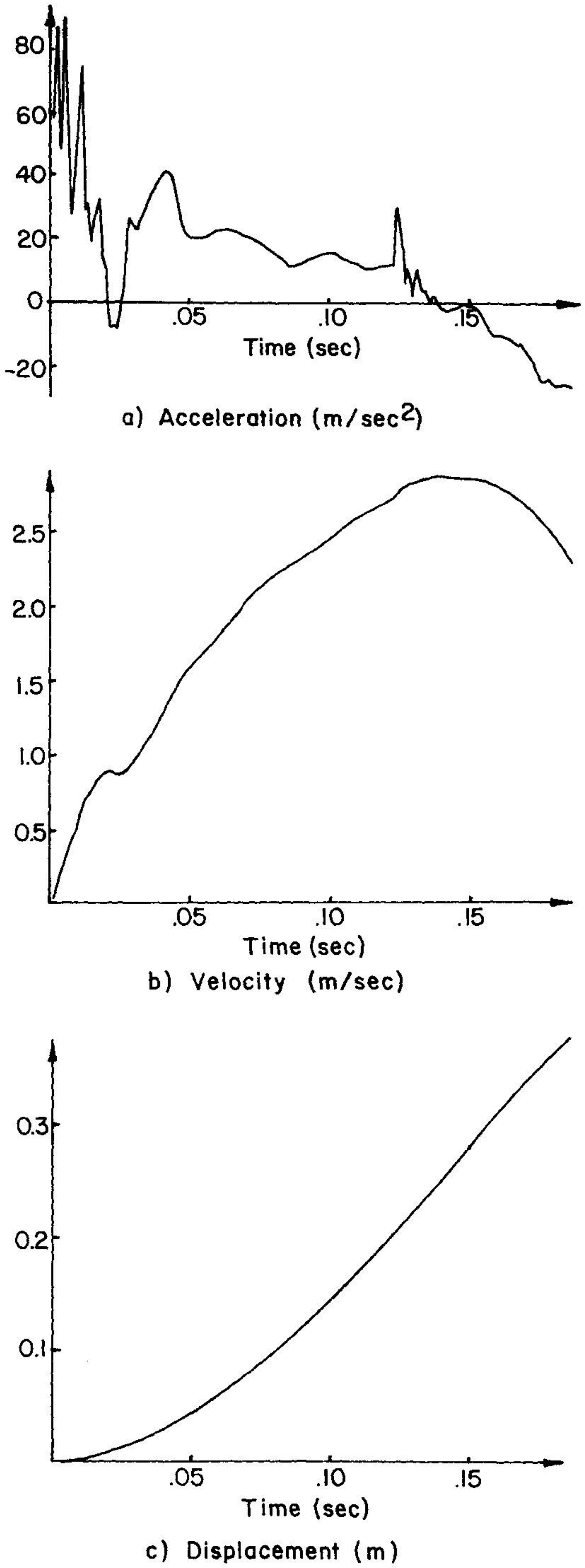


Fig.9 Time History Response of Tunnel Midspan Section

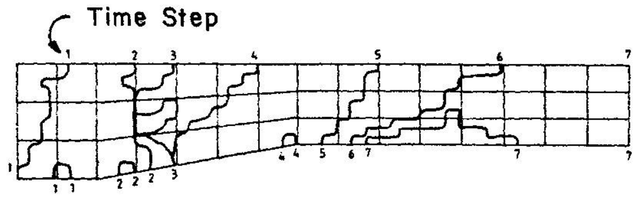


Fig.10 Propagation of Concrete Cracking Wave

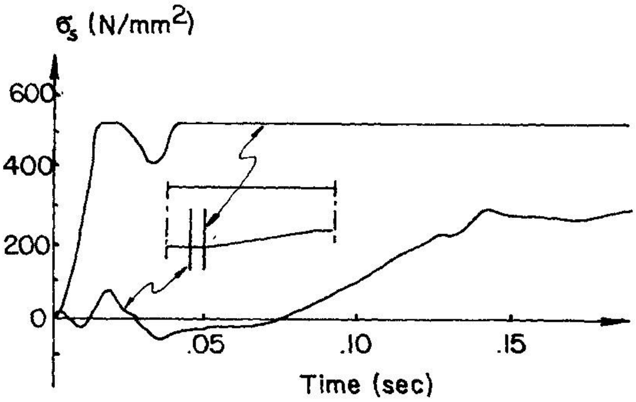


Fig.11 Time Histories of Stresses in Vertical Reinforcing Bars

2. MEYER, C. and OKAMURA, H., eds., "Finite Element Analysis of Reinforced Concrete Structures." Proceedings, U.S. - Japan Joint Seminar, Tokyo, ASCE, 1986.

3. BORST, R. DE et al., "DIANA - A Comprehensive but Flexible Finite Element System," Finite Element Systems Handbook, 3rd Edition, Springer Verlag, Berlin, Germany, 1983.

4. VAN MANEN, S.E. et al., "Example Calculations of Nonlinear Analysis for Reinforced Concrete Structures with DIANA," TNO-IBBC, Report No. BI-87-14/62.4.0000, TNO-IBBC, Delft, The Netherlands, 1987.

5. Collins, M.P., Vecchio, F.J. and Mehlhorn, G., "An International Competition to Predict the Response of Reinforced Concrete Panels," Canadian Journal of Civil Engineering (Ottawa), Vol. 12, No. 3, Sept. 1985, pp. 626-644.

6. MEYER, C., "Computer Analysis in Engineering Practice." Journal of the Boston Society of Civil Engineers Section ASCE, April 1983.

7. MEYER, C., ed., "Finite Element Idealization." ASCE, New York, 1987.

8. "Investigation of a Model of a Tunnel Section." TNO-IBBC, Report No. B-76-197/04.4.1080, TNO-IBBC, Delft, The Netherlands, 1976.

9. "Investigation of Transporting Dangerous Materials Through Tunnels," Rijkswaterstaat, Directie Sluizen en Stuwen, Utrecht, The Netherlands, 1982.