Dynamic analysis of aircraft impact using the linear elastic finite element codes FINEL, SAP and STARDYNE

Lundsager, Per; Krenk, S.

Publication date:
1975

Document Version
Publisher's PDF, also known as Version of record

Citation (APA):
Lundsager, P., & Krenk, S. (1975). Dynamic analysis of aircraft impact using the linear elastic finite element codes FINEL, SAP and STARDYNE. (Risø-M; No. 1817).

Title and author(s) Dynamic Analysis of Aircraft Impact Using the Linear Elastic Finite Element Codes FINEL, SAP and STARDYNE. Per Lundsager Steen Krenk	Date 1975.08.28
	Department or group Engineering
7 pages + tables + 9 illustrations	Group's own registration number(s)
	Copies to

Abstract

The static and dynamic response of a cylindrical/spherical containment to a Boeing 720 impact is computed using 3 different linear elastic computer codes: FINEL, SAP and STARDYNE. Stress and displacement fields are shown together with time histories for a point in the impact zone. The main conclusions from this study are:

- In this case the maximum dynamic load factors for stress and displacements were close to 1, but a static analysis alone is not fully sufficient.
- More realistic load time histories should be considered.
- The main effects seem to be local. The present study does not indicate general collapse from elastic stresses alone.
- Further study of material properties at high strain rates is needed.

Available on request from the Library of the Danish Atomic Energy Commission (Atomenergikommissionens Bibliotek), Risø, DK-4000 Roskilde, Denmark
Telephone: (03) 35 51 01, ext. 334, telex: 43116

ISBN 87 550 0356 7

Introduction

During the years the problem of aircraft impact on nuclear reactor containments has been the subject of considerable interest for a number of reasons, most of which have been discussed by Sütterlin [1]. The problem involves consideration of several aspects such as containment type, material model, impact characteristics and structural analysis method.

The aim of the present study is to compare the results of static and dynamic analysis of an idealized containment structure by comparing the results of 3 linear finite element codes, which at present are available to the Danish Atomic Energy Commission.

The containment structure. Figs. 1 and 2 show a cylinder with a spherical cap. The mean diameter of the cylinder is 36.5 m, the radius of the sphere is 28,4 m and the height of the cylinder is 59.3 m out of a total of 63.9 m. The wall thickness is 1.10 m with the exception of a heavier section just below the cap and the polar crane console shown in fig. 1.

The material model chosen for this study is the simplest possible: Isotropic linear material with $E = 3 \cdot 10^{10}$ N/m^2 and $\nu = 0.15$, which are representative values for concrete in linear analysis of quasistatic loading. This model, which excludes the effects of steel parts from this comparison, has been used also in the dynamic part of the analysis. The main reason for this is that no established dynamic concrete properties was found in the litterature.

The impact mechanism chosen is that of Riera [2], which has achieved common acceptance, as a number of applications shows, cf. Dietrich and Fürste [3] , Yang and Godfrey [4], Dritteler [5] . It is most often assumed, that the load may be considered as a pressure distribution over a specified constant area with a specified time history, and this approach is used here. The area, the load time history and the peak load depends on the aircraft type and velocity considered. It is obvious that a linear analysis using quasistatic material constants becomes increasingly dubious with shorter impact time. It is therefore assumed that the airplane is a Boeing 720 crashing at 200 knots giving an impact time of 0.33 sec, Riera [2]. This is the longest time history of those available in the open literature.

The structural analysis has been performed by use of three different linear finite element codes:

- FINEL of Burmeister & Wain [6]
- STARDYNE of MRI [7]
- SAP of Berkeley University [8]

Of these STARDYNE and SAP are rather conventional finite element programs using low order interpolation functions and a 3-dimensional mesh. The version of FINEL used in the present study has been developed specially for axisymmetric geometries. The mesh is 2-dimensional, only the cross section being modelled, fig. 1, and variations of displacements and stresses with the cross section angle are expressed in terms of truncated Fourier series. Thus the equations are decomposed in a number of smaller systems each related to a certain order in the Fourier series.

Representation of the structure

Due to the differences between the FINEL code and the two others, two different element models have been used. These models, including the loading patterns, were used in both the static and the dynamic analysis. The loaded elements are hatched on figs.1 and 2.

The FINEL model is shown in fig. 1, which is a computer plot of the axial section. The lowest ring element is a boundary element used to simulate a very stiff support. In the elements hermitean interpolation functions are used in the cross section plane. The displacements and their first order derivatives are used as node degrees of freedom, whereby stress continuity at the nodes is ensured.

The SAP and STARDYNE models fig. 2 use plane shell elements. In STARDYNE the triangular elements are constant stress membrane elements combined with the LCCT11 element, Clough and Felippa [9]. Four such triangles are used to form the square element. The SAP elements are analogues to this, the LCCT11 element being replaced by the LCCT9 element, [10].

The coarseness of the shell model is a result of the wish for more than one load case combined with the need for modest computing costs. The model is not expected to give detailed information of the stress distribution in the vicinity of the loaded area, and the SAP model may be stiffer than the STARDYNE model.

Static analysis

A total static load of $8 \cdot 10^7$ N was applied to the containment, and figs. 3, 4 and 5 show some results from the static analyses.

Fig. 3 is a computer plot of the deformed FINEL geometry. It is obvious, that the deformation mainly consists of a cantilever beam mode superposed by a local deformation at the load. The deformations from the shell model, which represents the middle surface of the structure,are so close to the deformed FINEL middel surface, that they are omitted.

The FINEL code gives the stresses at 16 evenly distributed points of the cross sections of each ring element, and it is therefore possible to give a very detailed picture of the stresses in the structure. As representative stresses the longitudinal and circumferential surface stresses of the cylindrical part of the containment were selected.

The axial section shown is the plane of symmetry with respect to the load. The curves do not show the stresses at the polar crane console because of the big differences in this area between the models adopted by the programs.

While it would be difficult in the scale of fig. 3 to separate the displacements of the three models from each other, this is certainly possible for the stresses. All three models agree well at some distance from the impact area, but especially SAP deviates notably in this area being more stiff than the others. The difference between SAP and STARDYNE is caused by the different bending elements used, this being the only difference between the shell models. Nevertheless the shell models are reasonably close on the FINEL results considering that the points are averages of elements in a coarse mesh.

It is obvious from the stress distribution that the load has mainly local effects. The tensile stress level of 5-10 N/mm^2 at the bottom of the cylinder is certainly high, but it can easily be taken care of by reinforcement and prestressing. This does not hold for the very high stresses at the load. The stress curves do in fact suggest the possibility that also a crashing airliner might lead to a penetration mode failure of the containment which is the normally anticipated failure mode in the case of a crashing fighter, Dietrich and Fürste [3].

The FINEL representation seems to be slightly softer than the shell representation. It should be so since it considers shear deformation, which the shell elements do not. On the other hand the horizontal resultant of the load acting on the impact area is slightly larger in FINEL than in SAP and STARDYNE. This is due to the truncation of the Fourier series representation in FINEL of the force, which will give small opposite forces in the vicinity of the impact area. Thus the local load is slightly more than intended for a given total horizontal load. It is therefore not possible to conclude how much the FINEL model is softer than the shell models.

Dynamic analysis

Also in the dynamic analysis the load was prescribed as a pressure, the time history of which has been derived under the assumption, that the impacted wall does not vibrate. The mass of the aircraft was thus in a certain sense neglected.

The time history of the force was applied in the form presented by Yang and Godfrey [4] and it is shown in fig. 6. The peak load $8 \cdot 10^7$ N is equal to the load used in the static analysis. This function is a simplification of the more realistic, but also more complicated time history presented by Riera [2], for a Boeing 720 at 200 knots. The duration of the impact is o.33 sec which is much longer than the 0.07 sec stated by Dritteler, Gruner and Sütterlin, [11] for a F5 Phantom at approximately 415 knots. This difference in duration was the main reason why the Boeing was chosen instead of the Phantom, since it was expected that the latter would demand a more detailed investigation due to more pronounced secondary dynamic effects.

In view of the static analysis it was considered as unnecessary and uneconomic to use both conventional codes in the dynamic analysis. This was therefore performed with FINEL and SAP only. After some tests time steps of 0.0050 sec. for FINEL and 0.0025 sec. for SAP were chosen. Both programs use the Wilson Theta method, Bathe and Wilson [12].

In order to compare the results from the programs so to say under their own conditions it was decided to normalize the dynamic results from each program by relating them to the corresponding results from the static analysis performed with the same program. This also gives a direct illustration of the deviation from a quasistatic calculation. Fig. 7 shows for both programs the normalized displacement history for a point within the impact area, and it appears that both curves follow the quasistatic response closely during most of the impact time. The peak of the displacement response occurs about 0.02 sec. after the peak of the forcing function. It should be noted that the maximum dynamic displacement in this case is only 0.9 times the corresponding displacement under static load.

The phase difference appearing from $t \sim 0.3$ sec. may be due to the greater stiffness of the SAP model.

The displacement curves in fig. 7 look very different from what would be expected from an undamped single mass model. This is explained by fig. 8, which shows the deformed shape of the cylinder as computed by SAP at various time points.

Immediately after the impact a local impression is formed. This impression becomes deeper until approx. t = 0.2 sec. At this time the deformed shape is very similar to that of the static case, but the displacements in the lower part of the cylinder are still increasing. At t = 0.3 sec. the displacement field is dominated by the outward motion of the console for the polar crane. It appears that the motion of the point considered in fig. 7 is influenced by at least four phenomena:

- a cantilever beam motion of the cylinder
- a bending wave proceeding downwards
- a local vibration of the impression
- the motion of the crane console

These partly secondary effects explains why a single mass model behaviour of the point of fig. 7 is not obtained, and for the containment as a whole a single mass model dynamic calculation combined with a static finite element analysis would give very poor maximum stress results.

Figs. 7 and 8 indicate that the maximum stress witnin the impact area should be expected at approx. t = 0.2 sec. This is confirmed by fig. 9, which shows the longitudinal stress at the outer surface as a function of time. The stress has been normalized in the same way as the displacements. The special conditions in the vicinity of the console due to its rather dominating motions are expected to be considerably dependant on the location of the load relatively to the console, and further study of this problem seems necessary.

Conclusions

Although the study presented above has by no means reached a final form, it seems reasonable to draw some conclusions of the work done while bearing in mind that it is based on linear, isotropic elastic analysis of a specific structure.

The maximum stress and displacement response of the impact area occurs at approximately 0.01 sec. after the maximum load. At this time both stress and displacement fields are very similar to the corresponding fields found by a static analysis. Furthermore the dynamic load factors for the impact zone are close to 1, the DLF for stress and displacement being 1.1 and 0.9 respectively. For points outside the vicinity of the impact zone the DLF's should be expected to differ considerably from 1, and they will occur at different times for different points.

The close resemblance between the dynamic and the quasistatic response in the impact zone suggests that more reliable and realistic load time histories should be considered.

Under the assumptions mentioned above the main effects of the impact seem to be local. This conclusion is based on both the static and the dynamic analysis, although the latter shows some secondary effects. The question of penetration contra general collapse seems to a certain extent to depend on these secondary effects e.g. bending waves and separate oscillation of the console. The present study does not indicate general collapse from elastic stresses alone.

The present results points more towards a closer examination of material behaviour at large strain rates than to a more refined conventional dynamic analysis. This is in agreement with a tendency in the litterature.

A general evaluation of the three finite element codes used here is not possible on basis of this study. The two conventional codes have done remarkably well considering that the Fourier expansion technique adopted in the present version of FINEL is ideally suited for the problem in the form treated here. It seems, however, that none of the codes in their present form are suited for the more detailed analysis suggested above. When using a shell model the limited ability of SAP to model large moment gradients should be kept in mind.

Aknowledgement

Mr. V. Hoppe, Burmeister & Wain Engineering Co. Ltd., Copenhagen has placed the FINEL calculations at the authors' disposal, which is gratefully aknowledged.

References

[1] L. Sütterlin : Über die Gefährdung von Kernkraftwerken durch äussere Einwirkungen.
IRS Fachgespräch, Köln, October 1974.

[2] J.D. Riera: On the Stress Analysis of Structures Subjected to Aircraft Impact Forces.
Nucl. Eng. Des. 8(1968)415-426.

[3] R. Dietrich, W. Fürste: Beanspruchung und Bemessung von Kernkraftswerksgebaüden bei den äusseren Einwirkungen Flugzeugabsturtz and Druckwelle".
Techn. Mitt. Krupp. Forschungs-Berichte.
Band 31, H3 (1973).

[4] H.T.Y. Yang, D.A. Godfrey : Structural Analyse of Aircraft Impact on a Nuclear Containment Vessel and Associated Structures.
Nucl. Eng. Des. 11(1970) 295-307

[5] K. Dritteler: Technish-Physikalische Modelle für äussere Einwirkungen und Ableitung der Lastannahmen.
IRS Fachgespräch, Köln, October 1974.

[6] V. Hoppe, P. Lundsager: The Burmeister & Wain FINEL Code, a FEM System with Hermitean Isoparametric Elements, with Examples of Applications.
Preprints 3. SMIRT, Paper M 2/3, London 1975.

[7] MRI/STARDYNE 3, Static and Dynamic Structural Analysis Systems. User Information Manual.
CYBERNET Service, Control Data Corporation Publication no. 86603500, 1974.

[8] K.J. Bathe, E.L. Wilson, F.E. Peterson, H.H. Dovey:
SAP - A Structural Analysis Program for Linear Systems.
Nucl. Eng. Des. 25(1973) 257-274.

[9] R.W. Clough, C.A. Felippa: A Refined Quadrilateral Element for Analysis of Plate Bending.
Proceedings 2nd Conference on Matrix Methods in Structural Mechanics.
Wright Patterson AFB Ohio 1968.

[10] K.J. Bathe, E.L. Wilson, F.E. Peterson:
SAP IV - A Structural Analysis Program for Static and Dynamic Analysis of Linear Structural Systems.
EERC Report No 73-11, College of Engineering, University of California, Berkeley June 73.

[11] K. Dritteler, P. Gruner, L. Sütterlin:
Zur Auslegung Kerntechnischer Anlagen Gegen Einwirkungen von Aussen. Teilaspekt Flugzeugabsturtz. Zwischenbericht IRS-W-7 (Dezember 1973).

[12] K.J. Bathe, E.L. Wilson: Stability and Accuracy Analysis of Direct Integration Methods.
Int. J. of Earthquake Engg. and Struct. Dynamics, Vol. 1, no. 2 1973.

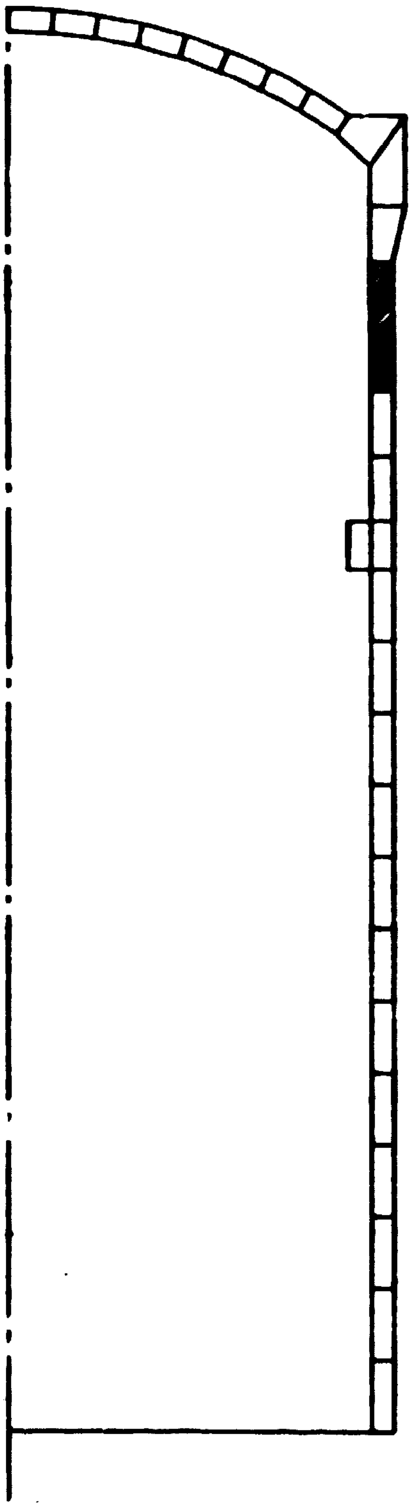

Fig. 1: The FINEL model. FINEL computer plot. Hatching shows the impact area.

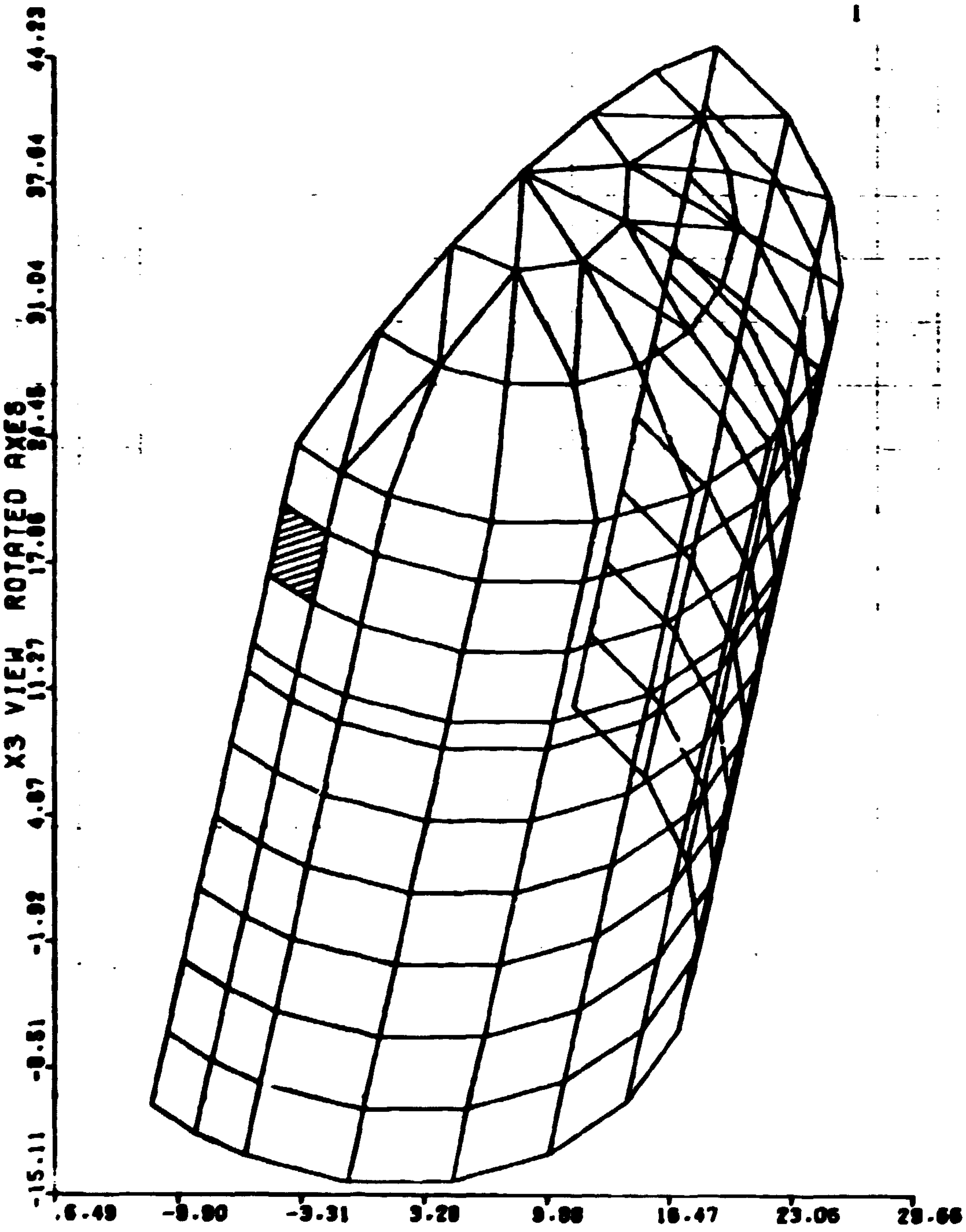

Fig. 2: The shell model. STARDYNE computer plot. Hatching shows the impact area.

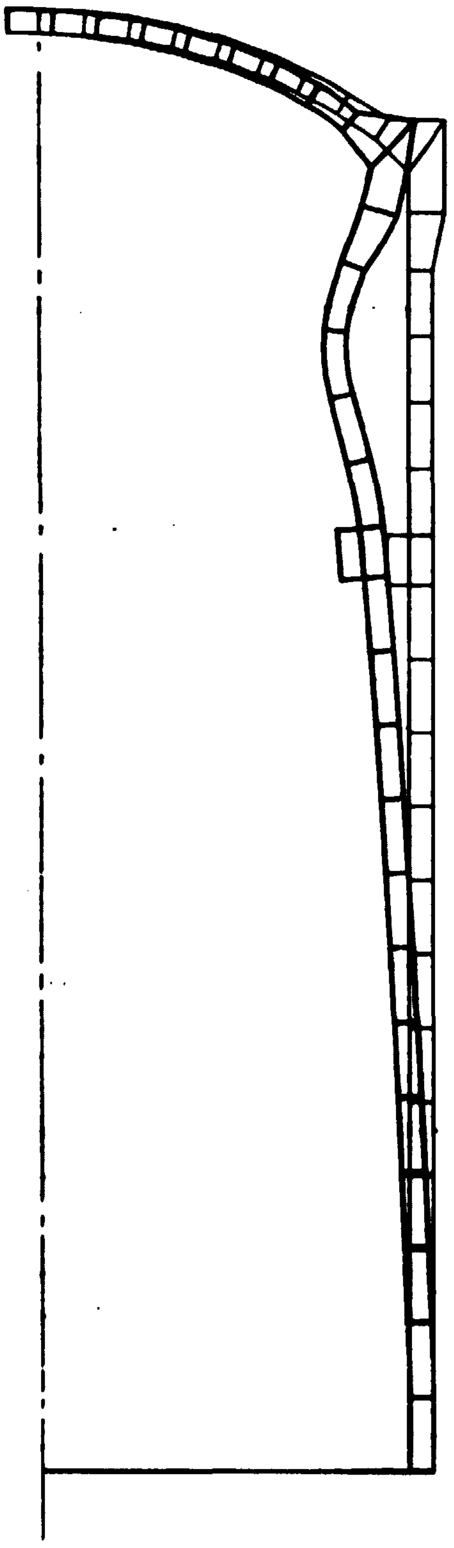

Fig. 3: Deformation due to static load. Actual displacement = 3.00 * shown displacement. FINEL computer plot.

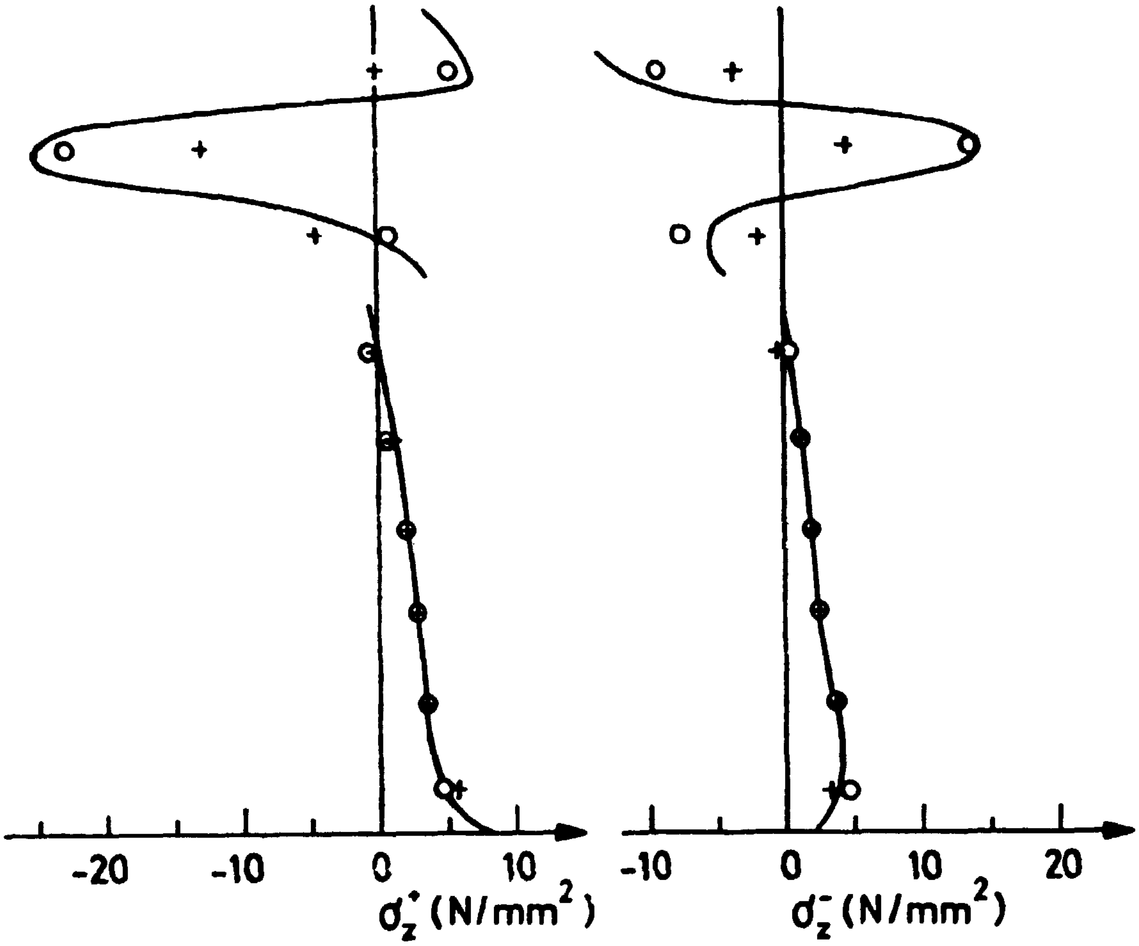

Fig. 4: Axial section through the midpoint of the impact area.

σ_z^+ = longitudinal surface stress at outer surface.

σ_z^- = longitudinal surface stress at inner surface.

- FINEL, O STARDYNE, X SAP

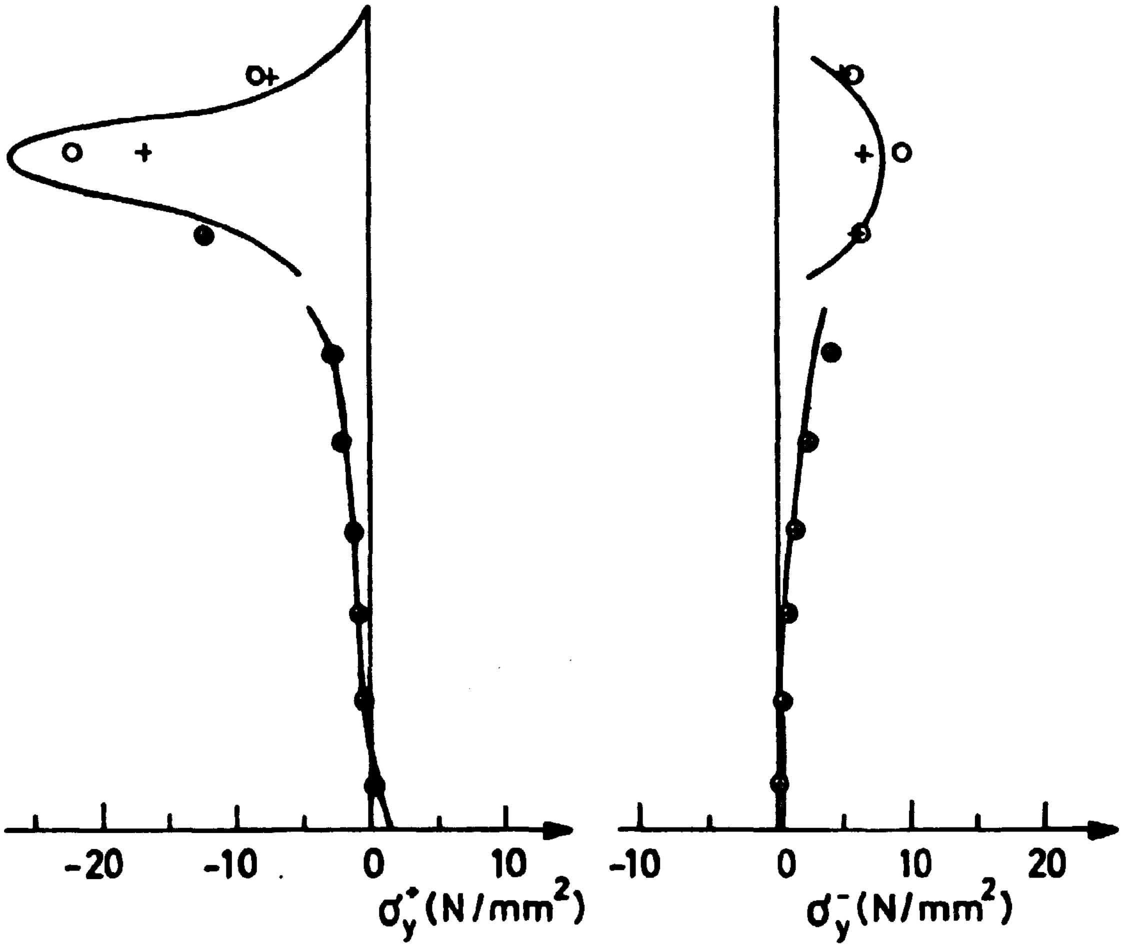

Fig. 5: Axial section through the midpoint of the impact ares.

σ_y^+ = hoop surface stress at outer surface.

σ_y^- = hoop surface stress at inner surface.

- FINEL, O STARDYNE, X SAP

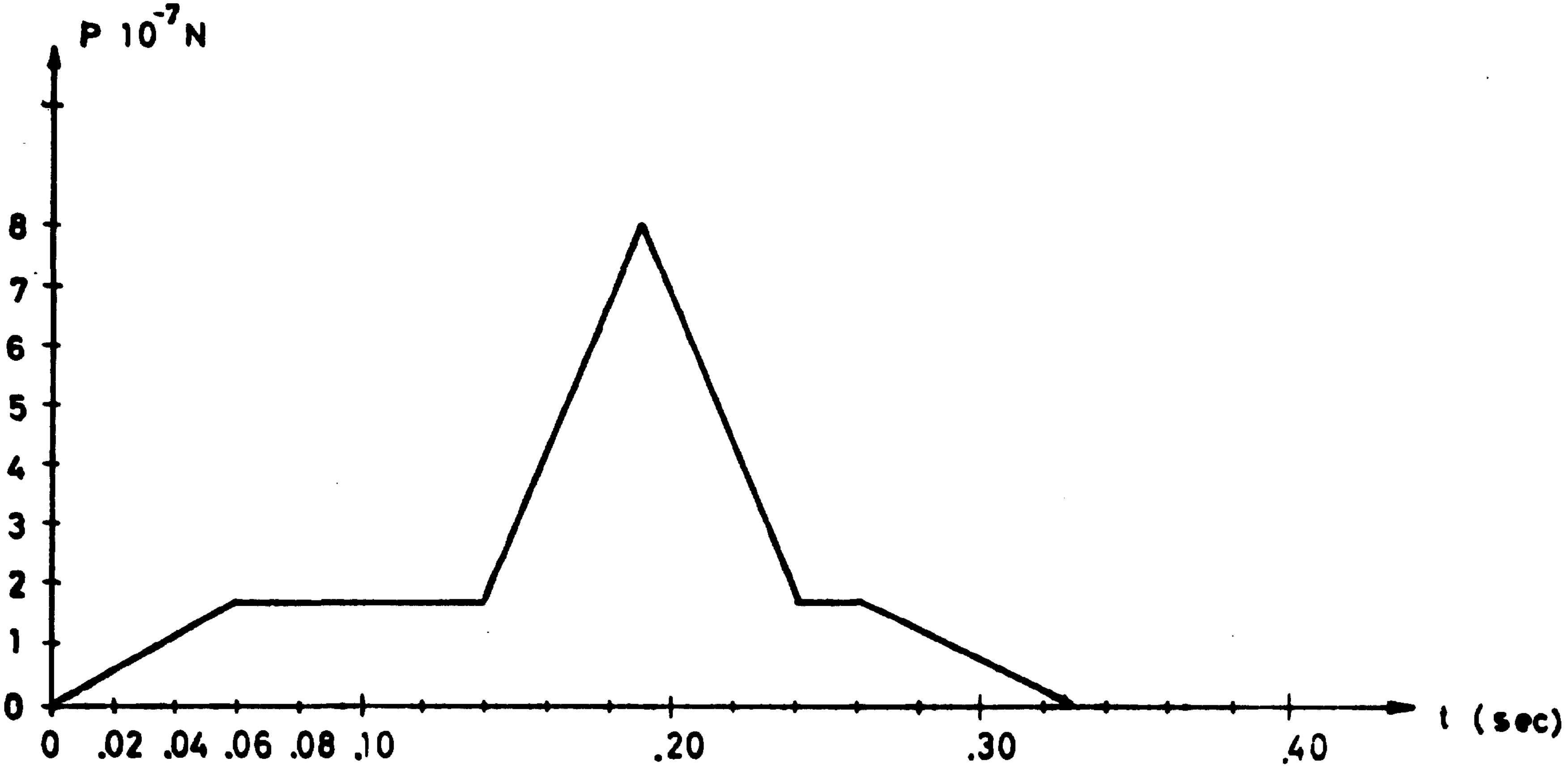

Fig. 6: Force time history for a Boeing 720 crashing at 200 knots, Yang and Godfrey [4].

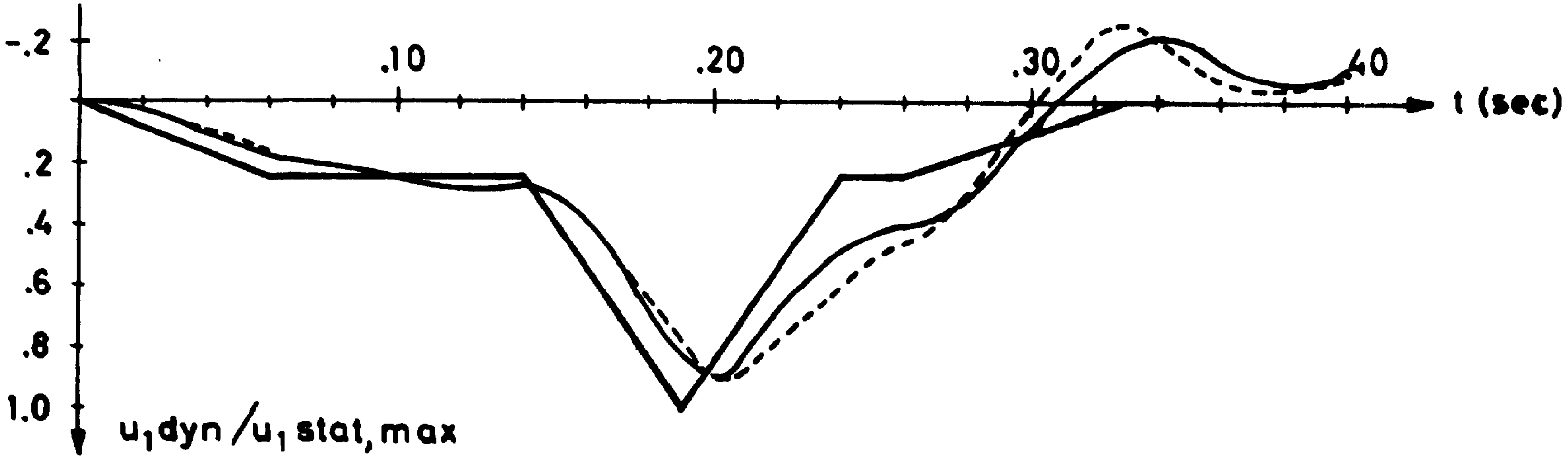

Fig. 7: Normalized displacement response of a point in the impact area.

u_1 dyn = displacement normal to the surface, dynamic analysis.

u_1 stat, max. = corresponding static displacement for maximum load.

- FINEL, --- SAP

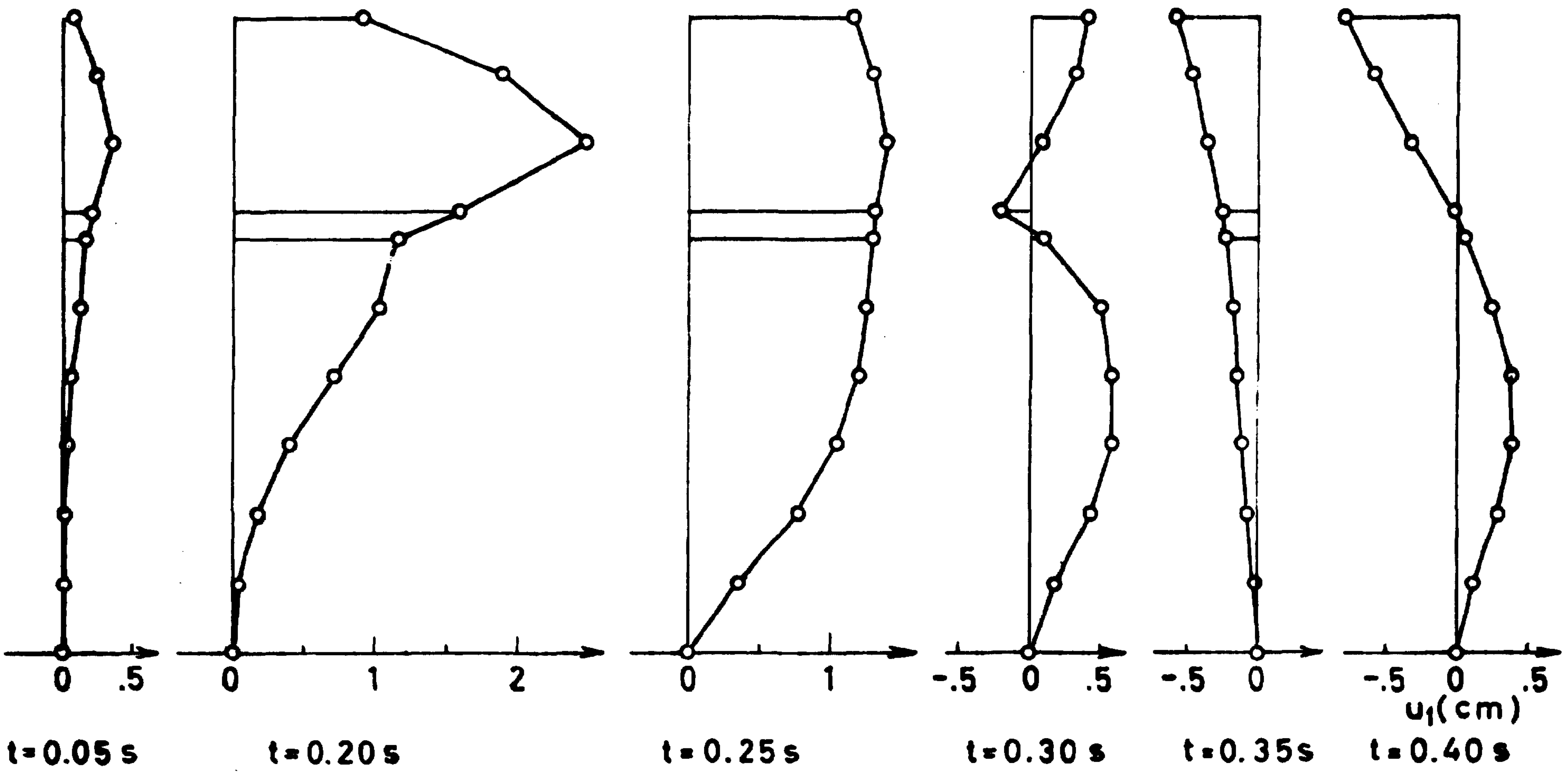

Fig. 8: Deformed shapes of the cylinder at various times. SAP results.

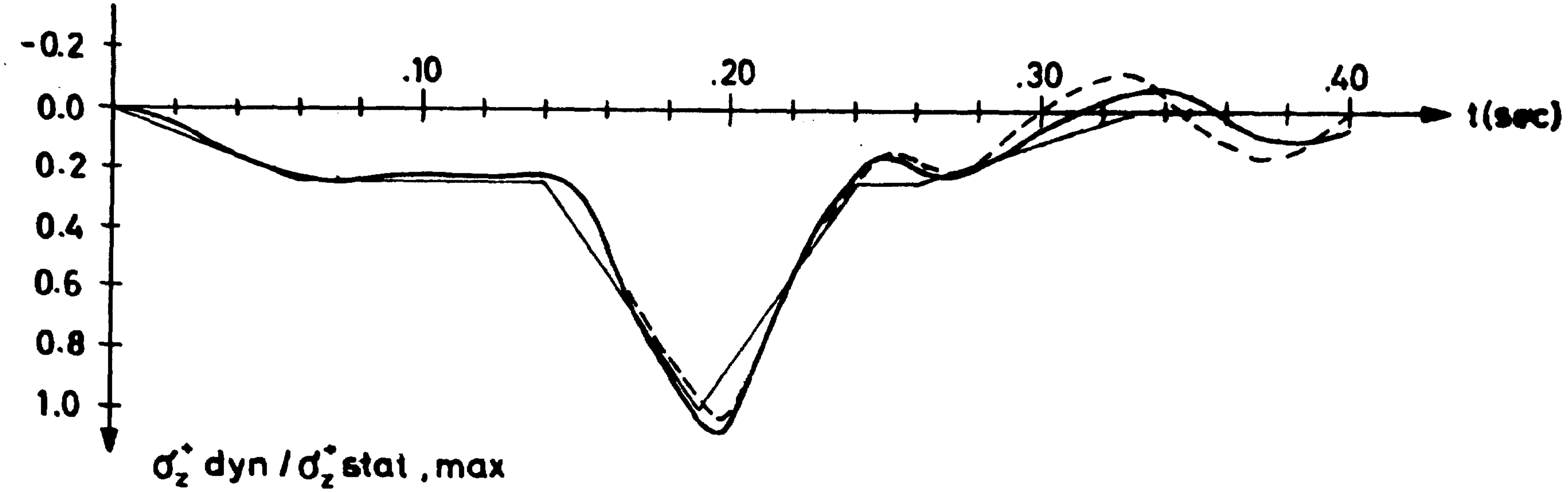

Fig. 9: Normalized stress response of a point in the impact area.

σ_z^+dyn = longitudinal stress at outer surface, dynamic analysis.

σ_z^+stat, max = corresponding static stress for maximum load.

- FINEL, ---- SAP