SOME USER EXPERIENCES WITH MSC/DYNA

by

Dr W.J.T.Daniel, Mechanical Engineering, University of Queensland

and

Dr Andrew Currie, Compumod Pty Ltd, Sydney

Abstract

The background to the explicit analysis techniques in MSC/DYNA is reviewed, and the inherent advantages and limitations of this approach, compared to the standard implicit MSC/NASTRAN formulation are discussed. Distinctive features of MSC/DYNA are reviewed. Examples of MSC/DYNA solutions to plate impact and penetration problems, rolling mill simulation, simulation of explosively formed particles, and modelling a golf club and ball are presented. These illustrate the graphics that can be obtained, and make use of the interfaces to PATRAN and MSC/XL and make use of input files from DYNA3D models. The software available to link MSC/DYNA to PATRAN and to MSC/XL is discussed.

1. Introduction

MSC/DYNA is a package primarily for the analysis of rapid, transient events such as impacts. While the program is associated with use of large models on supercomputers for vehicle crashworthiness studies, quite interesting and useful analyses can be done with limited computer resources. In this paper, some features of MSC/DYNA are reviewed and case studies presented of a number of MSC/DYNA models run successfully on a small MicroVAX II computer.

2. A Review of Features of MSC/DYNA

There are a number of ways in which MSC/DYNA is significantly different from other finite element software such as MSC/NASTRAN. Some of the more important differences will be outlined in this section, and their implications in particular analyses will be discussed in the case studies presented in section 3.

2.1 The Solution Process - What Controls the Timestep

In essence, each timestep, MSC/DYNA solves the equation

$$[M]\{\ddot{u}\} = \{F_{int}\} + \{F_{ext}\}$$

[M] is a diagonal lumped mass matrix, so that the equations can be regarded as explicitly giving accelerations $\{\ddot{u}\}$. These accelerations are found from internal forces $\{F_{int}\}$ and external or applied forces $\{F_{ext}\}$. The central difference method is used to predict a new set of displacements from the current accelerations.

$\{F_{int}\}$ may be written as $[K]\{u\}$ if material behaviour is linear and the displacement is small. However, calculations can be done at element level and the full [K] matrix need not be assembled. Internal forces on each element are found using nonlinear relations. Note that in any one time step, a displacement applied at one node can only lead to $\{F_{int}\}$ forces at nodes of elements sharing that node. Hence, in one timestep, a disturbance cannot propagate a distance any greater than the shortest element sidelength l. The timestep chosen automatically by MSC/DYNA is related to l/c, where c is the speed of sound present in a model. This timestep is short, typically microseconds or less, but the computation per timestep is low, because the accelerations are found explicitly. In the presence of severe nonlinearities, the low timestep is not a disadvantage, as the nonlinearities require a small timestep for convergence. The user does, however, have to be wary of inadvertantly placing some nodes too close together. The timestep can also be related to the highest natural frequency of the model, which is often associated with a wavelength comparable to l.

Any stiff spring may control the timestep. For this reason, regions of a model that are much stiffer than the rest are best modelled as rigid bodies. The default timestep has been found to work well, in fact the package is remarkably robust. A large timestep will lead to an unstable solution that rapidly diverges. Oscillating accelerations have been noted immediately after an impact, but these are brought under control by the bulk viscosity that is present in every model to avoid high frequency oscillations.

To solve problems that are quasi-static or which must approach a steady state dynamic situation, a large number of time steps becomes necessary. While this is not generally advisable, the ability of MSC/DYNA to handle material, geometric and contact nonlinearities may make such a choice the only viable one. A static solution can be obtained by using high damping (called dynamic relaxation). This must be used with caution, as the dominant modes of vibration must be close to critically damped to obtain convergence to the correct solution.

Note, by comparison, that time integration in most finite element packages, including MSC/NASTRAN, is done using implicit methods, in which a large set of equations is solved each time interval, much as in static analysis. Hence one node can influence other remote nodes in a single time step and the time step can be made much larger in some problems. However, all such implicit algorithms have difficulty with severe nonlinearities, which are treated with ease in MSC/DYNA.

2.2 Meshing

Whereas in static, linear elastic analysis the analyst can anticipate stress concentrations and refine a mesh locally, with MSC/DYNA the mesh must be made relatively fine and of relatively uniform density. There are a number of reasons for this.

(a) Immediately after an impact, the deformation is local to the region impacted. Stress waves spread and reflect, leading to ever changing stress levels corresponding to vibration that may be quite slowly damped. These processes must be modelled.

(b) The frequency response of the numerical model depends on the element size. Elements that are too large act rigidly at high frequencies and will reflect high frequencies in stress waves. Elements that are too small limit the timestep.

(c) Some materials in MSC/DYNA can fail, in the sense that the element no longer produces any loads on the model. For instance, tearing of a plate can be represented this way, as long as the mesh is fine enough so that deletion of elements is an adequate model of a crack. Clearly, this can only be done with very ductile materials, since the crack tip energy causing failure is distributed over the volume of the element.

2.3 Contact Mechanics

Surfaces that may contact are described by contact segments. During data preparation, these may be thought of as a layer of plates on a surface. A contact surface on a plate is in fact meshed as two sets of plate elements superimposed, sharing the same nodes. Typically contact surfaces are classified as a master surface and a slave surface. Every time interval, slave surface nodes are searched to determine if they are present in a volume normal to a master segment. They are checked for penetration of the master segment and restrained by stiffness terms and frictional forces as necessary. By this means, intermittent contact, or contact with gross sliding can be modelled. However, use of too many contact segments will slow the solution significantly.

Other simpler contact situations can also be modelled, for instance the "rigid wall" that cannot be penetrated by a set of nodes.

2.4 Rigid Bodies

A group of elements can be declared to form a rigid body, whether or not they are actually connected. This is convenient for reducing the size of a model, as a rigid body is represented internally as six equations of motion, loaded by forces on the centre of mass and moments about the centre of mass, found from the loads at nodes on the rigid body. The user needs to be careful with rigid body constraints since in MSC/DYNA Version I, a prescribed time history of velocity cannot be specified. A steady velocity can be achieved by using an initial velocity and a large mass. Similarly, a steady angular velocity can be achieved with a large moment of inertia. A feature of data preparation that may surprise the user is that a rigid body is given a Young's Modulus and a Poisson's ratio. These are used to describe the stiffness of any contact surface placed on the rigid body and should not be too high. Where contacting surfaces are of greatly differing stiffness, the stiffnesses are automatically rescaled at the start of the solution.

2.5 Loading

The elements in MSC/DYNA are underintegrated for the sake of efficiency in evaluating the internal forces, and are stabilized against zero-energy or hourglass modes of deformation. The User's Manual warns that a point load on one node may excite spurious hourglass deformation if the model is not properly constrained. This situation is hard to avoid. For instance, a curved surface impacting a flat surface will naturally first meet it at one point. However, these situations were found to be dealt with successfully in the models analysed.

2.6 Features Yet to be Implemented

MSC/DYNA is a commercial version of DYNA3D. Extra material models are present in DYNA3D, which make use of equations of state. One interesting material model is number 8 which models the effects of an explosion by causing elements to expand in volume dramatically as a shock wave passes through them. These extra material models, and other DYNA3D features, can currently be accessed with some difficulty in version 1 of MSC/DYNA.

2.7 Graphical Interfaces

Most of the work described here was done using PATRAN as a pre and post processor. A PATRAN model is translated using PATNAS to MSC/NASTRAN format. Extra MSC/DYNA data statements need to be added to this file. For convenience, the geometric data obtained from PATRAN can be left in one file and an INCLUDE statement used to incorporate property data from another file. With Version I, CQUAD4 statements representing contact segments must be edited to become CSEG statements. Output from a binary result file is translated to PATRAN neutral file format using DYPAT. Time history output can be plotted using DYNAXY, or alternatively PATRAN/PPLOT, in which case DYPPLOT is used to extract the data from a MSC/DYNA history file.

MSC/XL can be used instead of PATRAN, and has the advantage of producing data in MSC/NASTRAN format directly. To access the MSC/DYNA result file, a program called DYNA2XL can be run. Displacements, velocities and accelerations are then available for postprocessing in MSC/XL. Element quantities, such as equivalent plastic strain are also available, although which column of numbers is which is presently undocumented.

3. Case Studies

The following case studies were conducted as demonstration problems for clients of Compumod. Their technical assistance is acknowledged. The generous assistance of MSC's Mr Roger Keene is also acknowledged.

3.1 Simulation of a Rolling Mill

After initial contact, this is a steady state situation, and as such is demanding of MSC/DYNA, requiring a large number of time steps. The quarter symmetry mesh used is shown in Figure 1. The roller is modelled as rigid plate elements, constrained by a plate giving an elastic link to the axis of rotation. It was given a large moment of inertia to enforce a peripheral roll velocity of 5 m/s during the simulation. The steel slab was modelled using material 12. Detail results for stress, strain and motion were stored at many time steps. A steady state situation was found to develop soon after the billet had entered the rolls, giving typical deformation as in Figure 2, plastic strain as in Figure 3, and vertical stress as in Figure 4. The MicroVAX II CPU time to reach this steady state was about 10 hours. The inability of MSC/DYNA Version I to directly output contact forces meant that the vertical near surface stress had to be used for this purpose. Initial results have been very encouraging and rolling of steel sections is being attempted.

3.2 Impacted Plates

Three models of impacted plates with clamped boundaries were studied.

(a) A quarter model of a plate impacted near its centre. (Figure 5)
(b) A half model of a plate impacted near the midpoint of a longer edge. (Figure 6)
(c) A plate with two stiffeners impacted near one stiffener (Figure 7)

The impactor had a spherical surface and was modelled as a rigid body using solid elements. In doing this, it was necessary to avoid pentahedrons, as in version 1 of MSC/DYNA, a triangular contact segment does not recognize a triangular side of a penta element. Some quite distorted elements were used, but this is not a problem as they are rigid. The impactor was given an initial velocity of 154 m/s. The plates were modelled with material 24, which allows a piecewise-linear fit to the elastoplastic properties of the aluminium used, and also allows failure to occur. This was assumed at an equivalent plastic strain (log strain) of 18%. No allowance was made for effects of temperature or strain rate. Restarts were used every 0.5 ms to examine the state of the model and to delete failed elements. In the case of a plate symmetrically impacted, the plate initially dishes under the impactor, the strain distribution reflecting the shape of the impactor. However, by 1.0 ms, the plate has begun to tear, not around the impactor, but along the fixed edge closest to the impact. By 1.5 ms, this crack has grown notably, and a second one has begun along the edge of the impactor at one plane of symmetry. This is joined by a second crack starting at the other plane of symmetry and at the edge of the impactor. Shortly after 2 ms a large section of plate breaks off (Figure 8).

An alternate mesh was also used for this problem in order to check if the solution was dependent on the design of the mesh. The alternate mesh is shown in Figure 9, and is definitely too coarse along the edges. This was done on purpose to see if the failure zone would change to near the impactor. Elements were made radial and tangential to the impactor, to see if this would affect the direction of crack growth. A finer mesh was used on the impactor. Despite these changes, the resulting failed plate was similar to that from the other mesh (Figure 10).

The force of impact acting on the plate, was obtained from the acceleration of the impactor, and is shown in Figure 11. The first spike in acceleration immediately after impact are presumably due to the numerical model.

The plate impacted near one edge started to tear along the edge before 0.5 ms and also to tear along the plane of symmetry. The stiffened plate shows similar behaviour, the plate tearing along the stiffener, as shown in Figure 12. Typical CPU times on a MicroVAX II for 0.5 ms of simulation were 48 minutes for the centrally impacted plate, 36 minutes for the plate impacted near one edge and 91 minutes for the stiffened plate model.

3.3 Mass-Focussed Projectiles

A model of an explosively formed projectile was set up using PATRAN, as shown in Figure 13. The model consists of a dished copper plate, represented with two layers of solid elements, resting initially on a mesh of solid elements representing the explosive. Contact segments describe the interface. In order to allow the explosion to be unsymmetric, 180 degrees is modelled. After the explosion, a shock wave propagates from the detonation point, as can be seen in Figure 14, which shows a similar model, supplied by a client. The explosive elements expand greatly in volume. They exert pressures on the copper elements which become insignificant after about 50 microseconds. At this time, the model size can be reduced by deleting explosive elements. The copper elements deform due to their own inertia. Figure 15 shows the shape that the copper plate has assumed at 40 microseconds, its initial curvature having been reversed. At 60 microseconds, further deformation has occurred, as seen in Figure 16. This process continues, resulting in the copper crushing into a small projectile. 60 microseconds of simulation with this model, including the explosive, took 3.6 CPU hours on a MicroVAX II.

3.4 A Flexible Golf Club

Metal golf clubs are designed as a wedge of stainless steel. This study investigated possible flexible designs, such as that shown in Figure 17, without the handle. The club consists of two ends meshed with solid elements, connected by a flexible plate, the plate elements overlapping the solid elements. Meshing a golf ball with solid elements required care to avoid placing nodes too close together. For instance, if the Delaunay meshing option of MSC/XL is naively applied to a volume of one eighth of the ball, a ring of very closely spaced nodes is created near the axis about which a surface is swept to create a volume. Nodes need to be graded to keep them away from this axis, as otherwise an unacceptably small timestep results. The design of Figure 17 flexes as in Figure 18. This mode of vibration absorbs kinetic energy which is then denied to the ball. Hence the predicted performance is not impressive. If the two ends are restrained to be part of one rigid body, much better performance is obtained. This case gives time histories such as Figure 19, plotted with DYNAXY. The curves closely resemble the behaviour expected from a simple model of two masses joined by a spring, representing the stiffness of the ball.

This model, illustrated in Figure 20, ignores the handle of the golf club. Hence no external forces act on the system and momentum is conserved. The centre of mass of club and ball will move at a constant velocity v, given by the expression in the diagram. Until the ball separates from the club, they both oscillate in velocity about this steady value v. Since the ball started at zero velocity, it acquires a velocity of 2v as it leaves the club For the case of Figure 18, v = 42.5 m/s due to an initial velocity of 50 m/s of the club. It can be seen that the theoretical value of 85 m/s is not achieved by the golf ball, largely due to modes of vibration of the ball that are excited by the impact and make less energy available for translation.

To estimate a Young's modulus for the golf ball, by comparison with a nonlinear load deflection curve for compression of the ball obtained experimentally, a dynamic relaxation was conducted. The ball was grossly deformed between a rigid plate, with associated contact surface and a rigid wall. The resulting deflection was compared with experiment. It was found necessary to relax the default tolerance used to test convergence in MSC/DYNA to obtain a solution.

0.5 ms of simulation with model takes 7 hours of CPU time on a MicroVAX II. This could be reduced by use of symmetry. Symmetry was not exploited to give the option of including the handle of the golf club in the model.

4. Conclusions

MSC/DYNA is a robust and versatile package. While it presents the user familiar with other finite element software with quite a number of new concepts, it is not difficult to master.

As the above examples show, it can be applied to a wide range of problems, in addition to the more well-known problem types such as vehicle crash simulation. Additional capabilities, present in the parent program DYNA3D, such as the explosives modelling demonstrated above, should appear in future versions. It is noted that version 2 will contain some welcome additions, such as contact force printout, which will provide the user with more information.

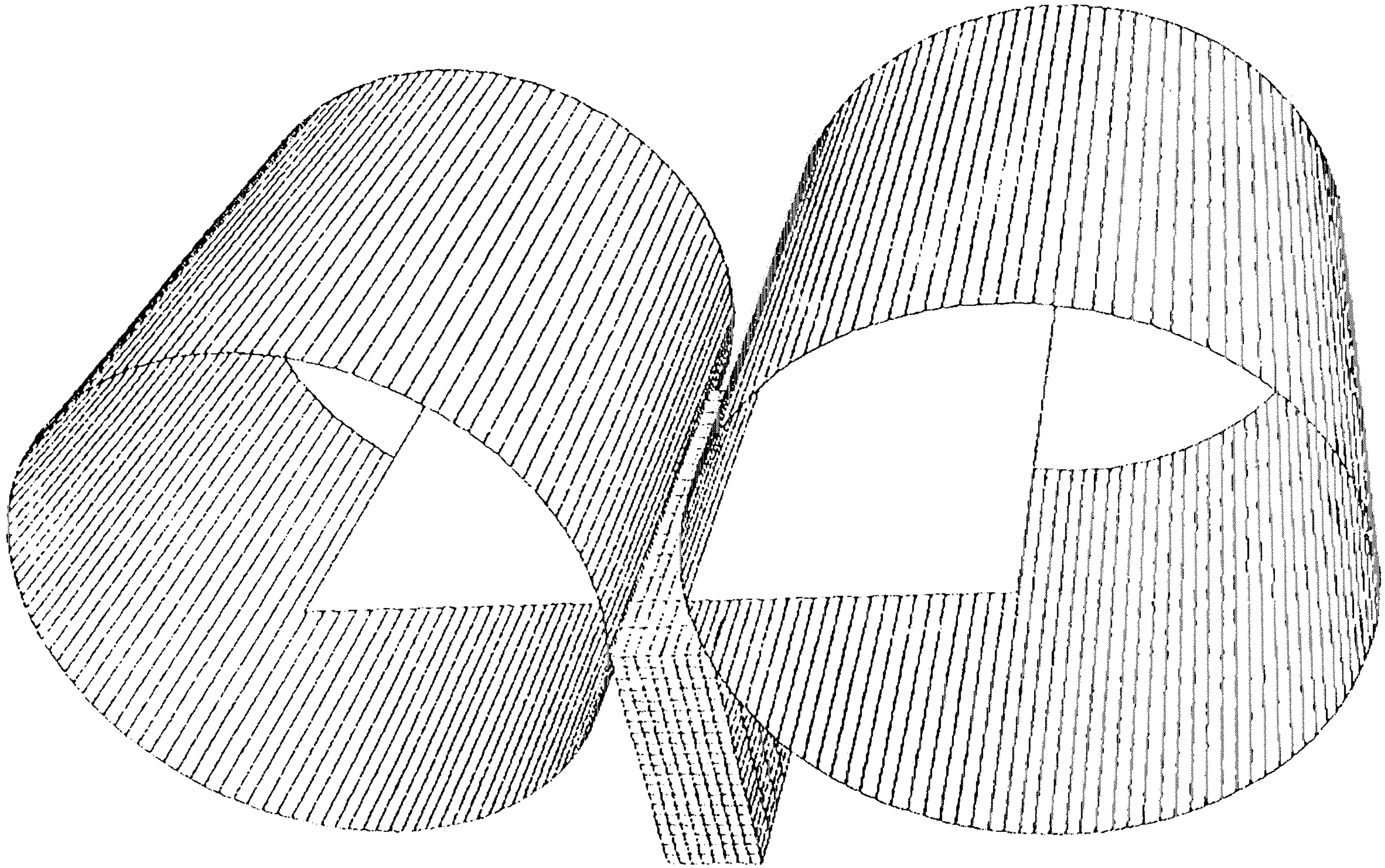


Figure 1: Model of billet and rolls

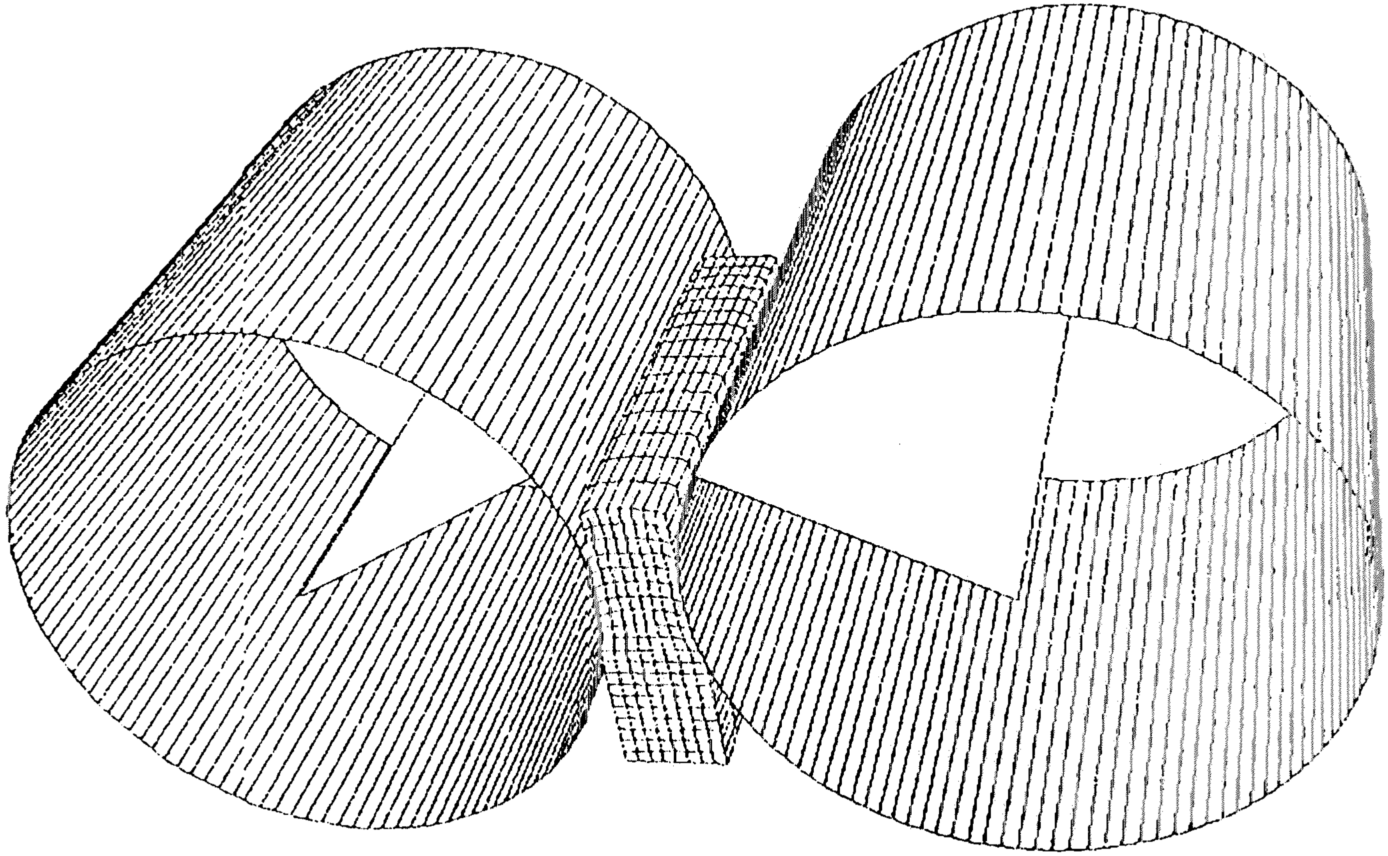


Figure 2: Deformation of the billet under the rolls

Figure 3: Steady plastic strain distribution in the billet being rolled

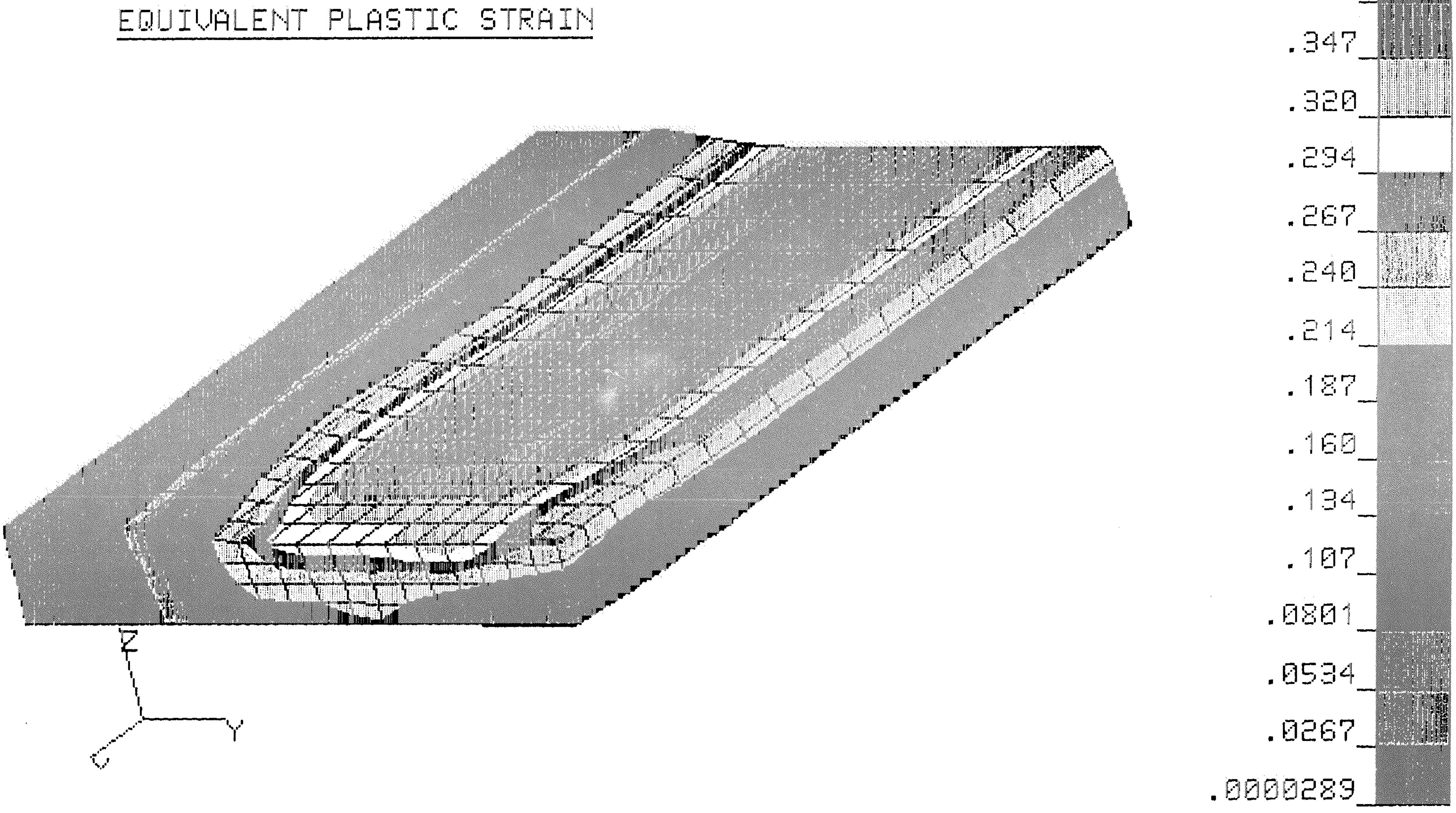
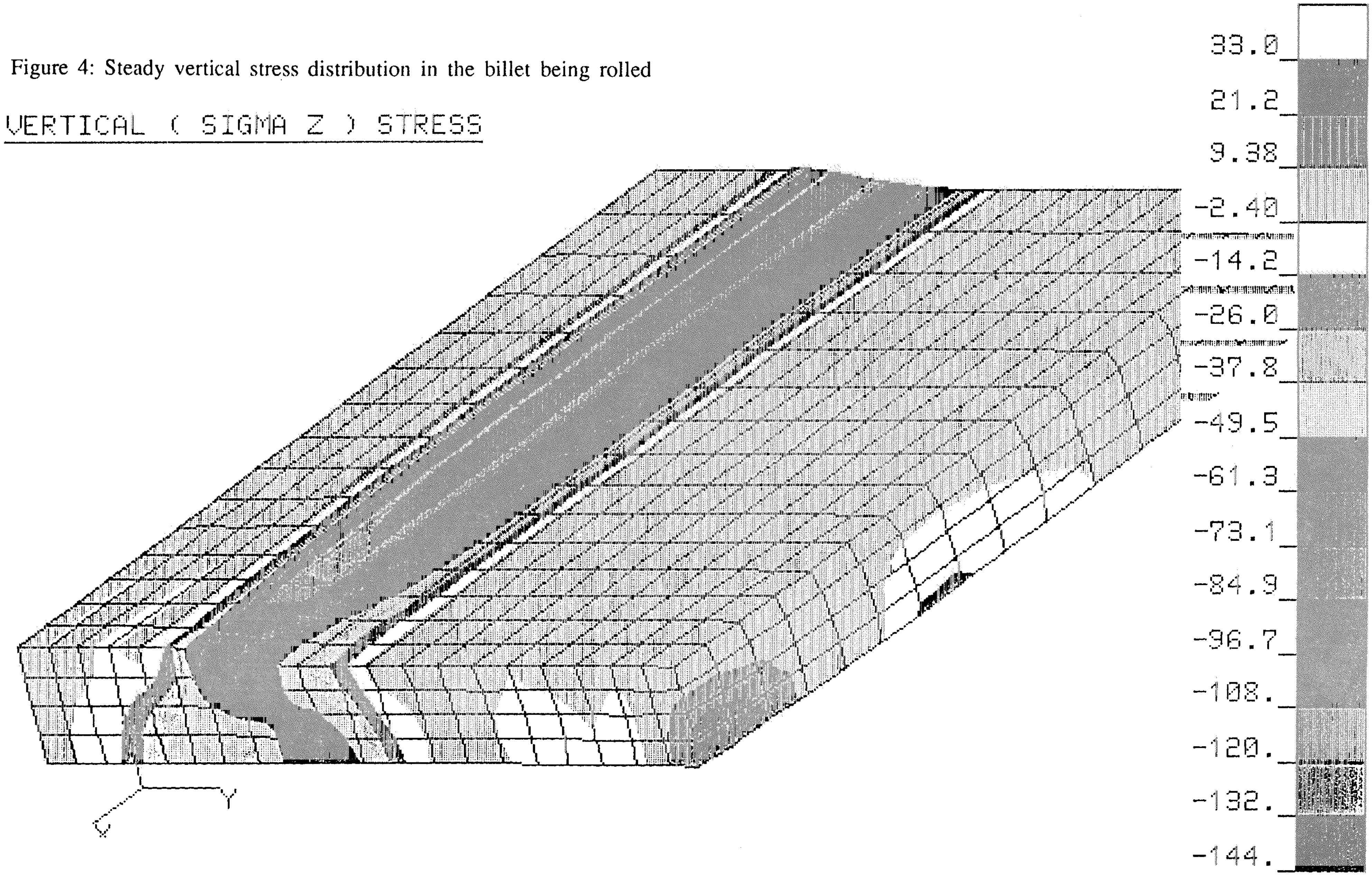


Figure 4: Steady vertical stress distribution in the billet being rolled



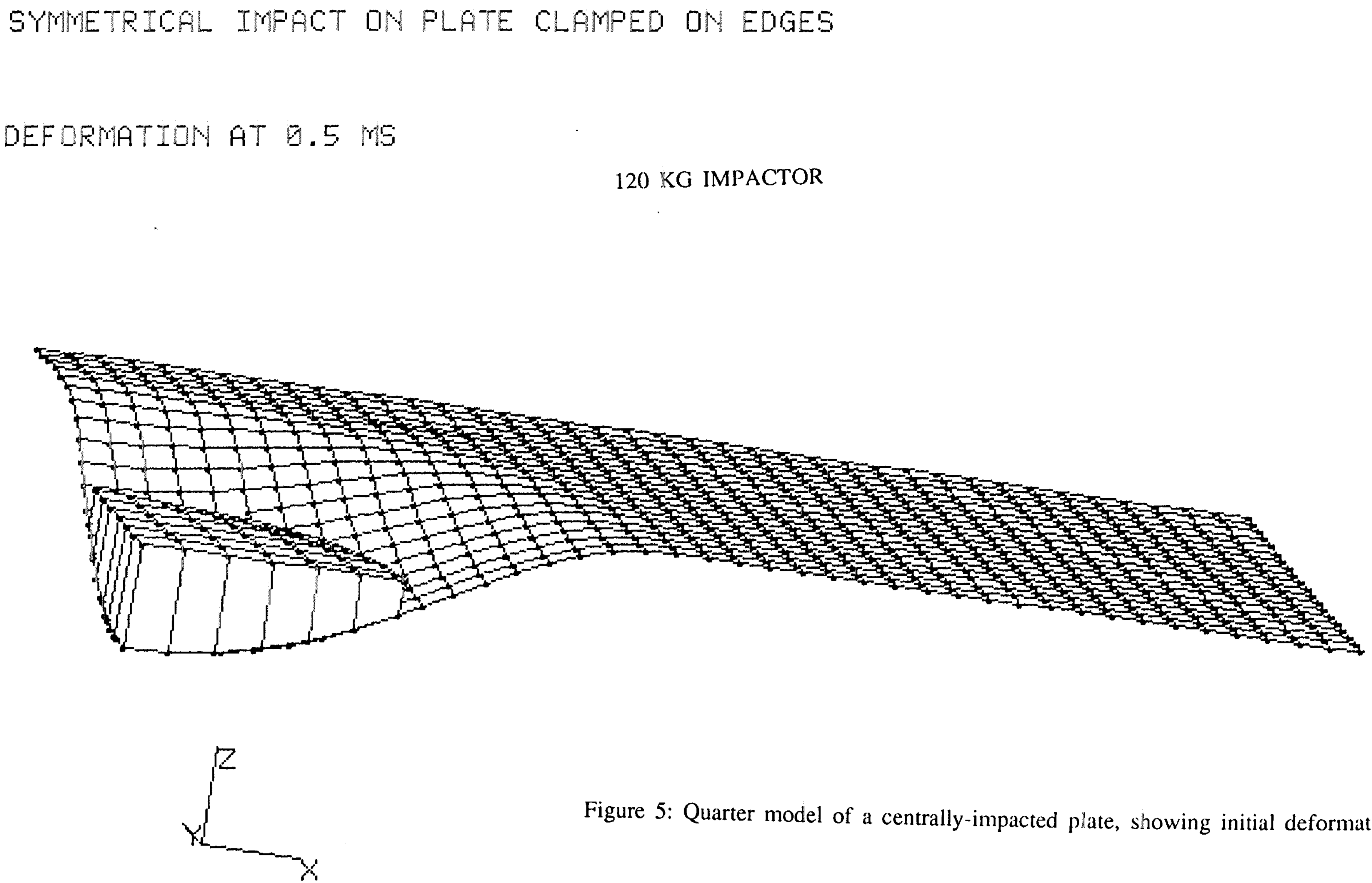


Figure 5: Quarter model of a centrally-impacted plate, showing initial deformation

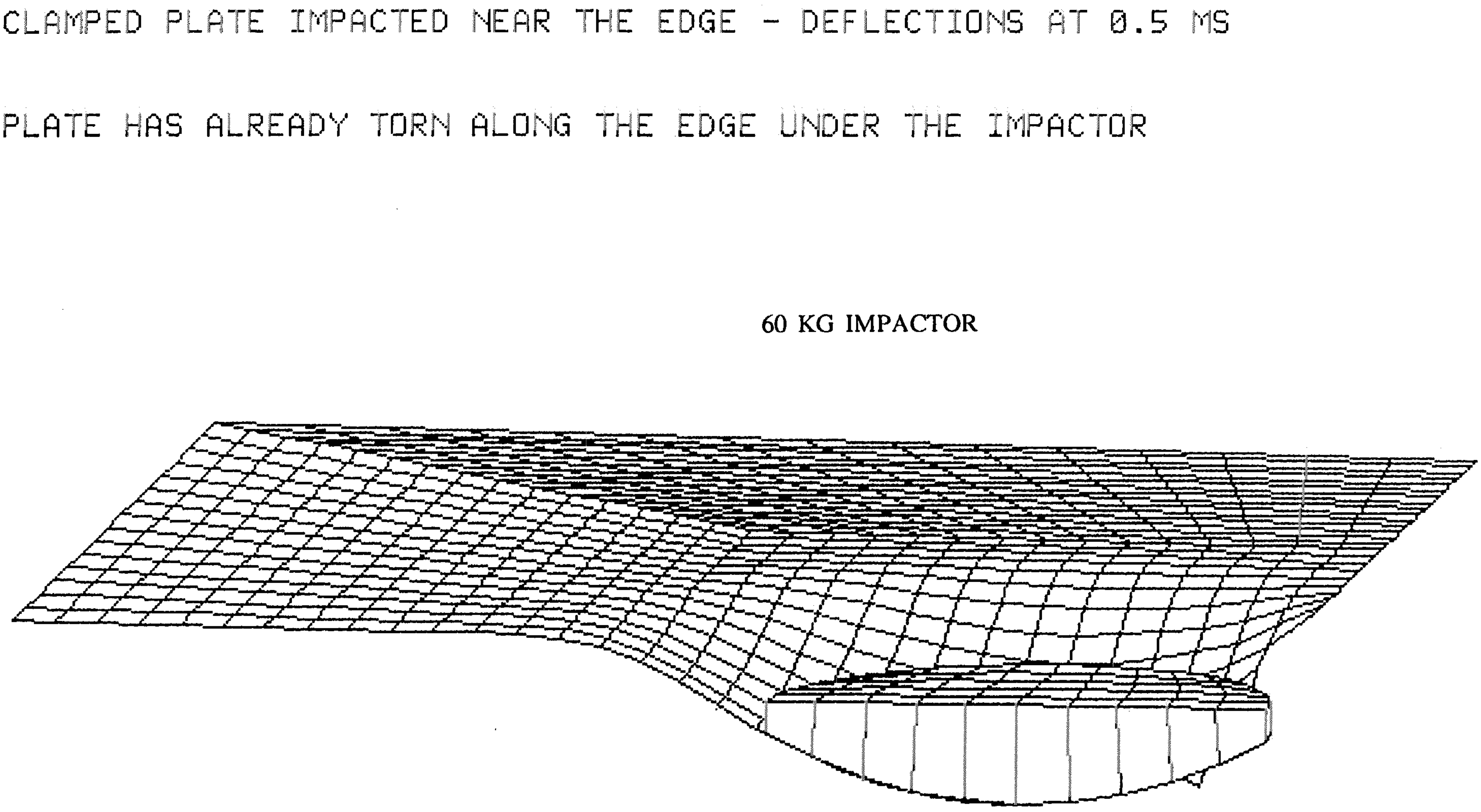


Figure 6: Half model of a plate impacted near one clamped edge, showing tearing along edge impacted

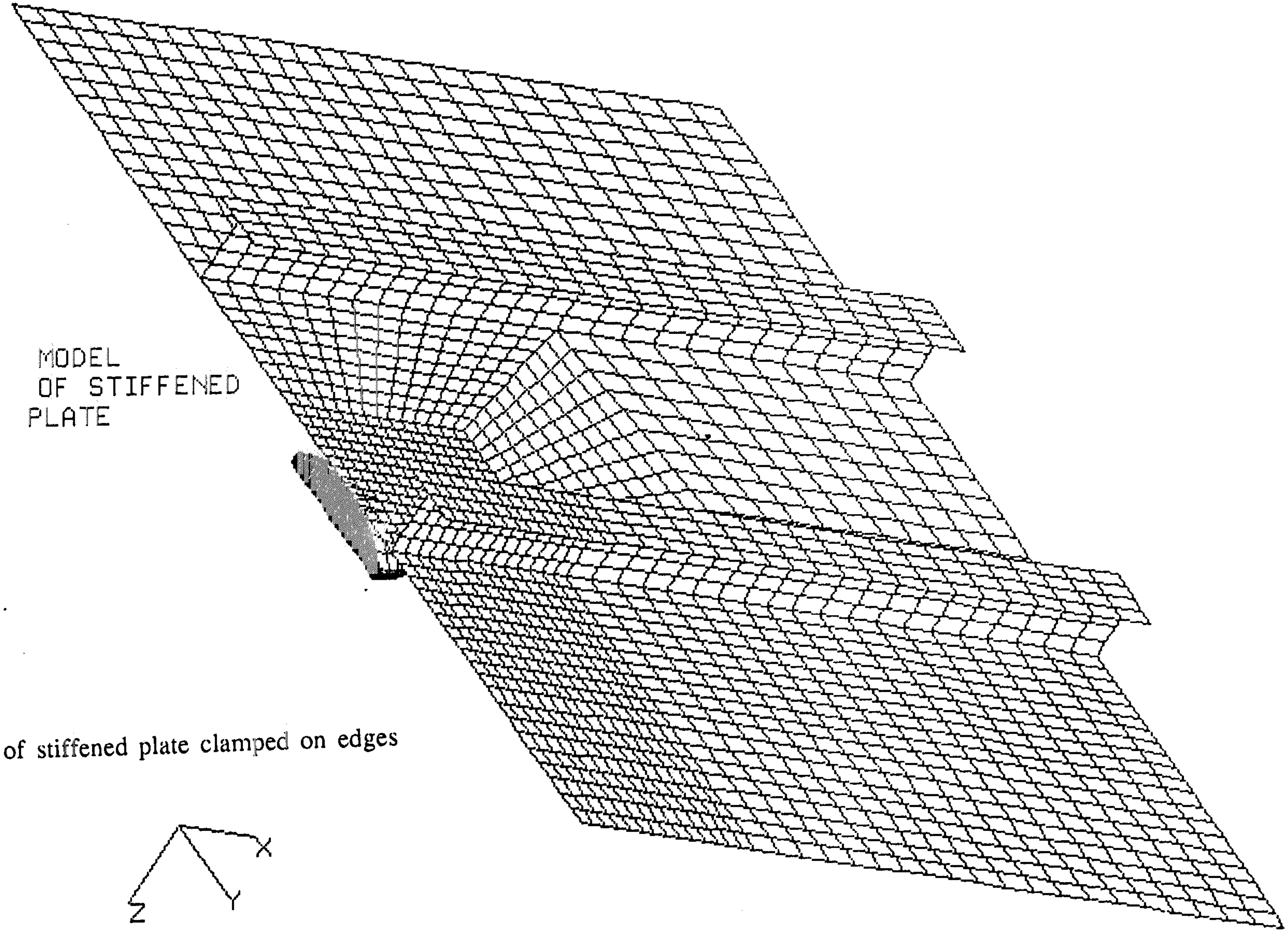


Figure 7: Half model of stiffened plate clamped on edges

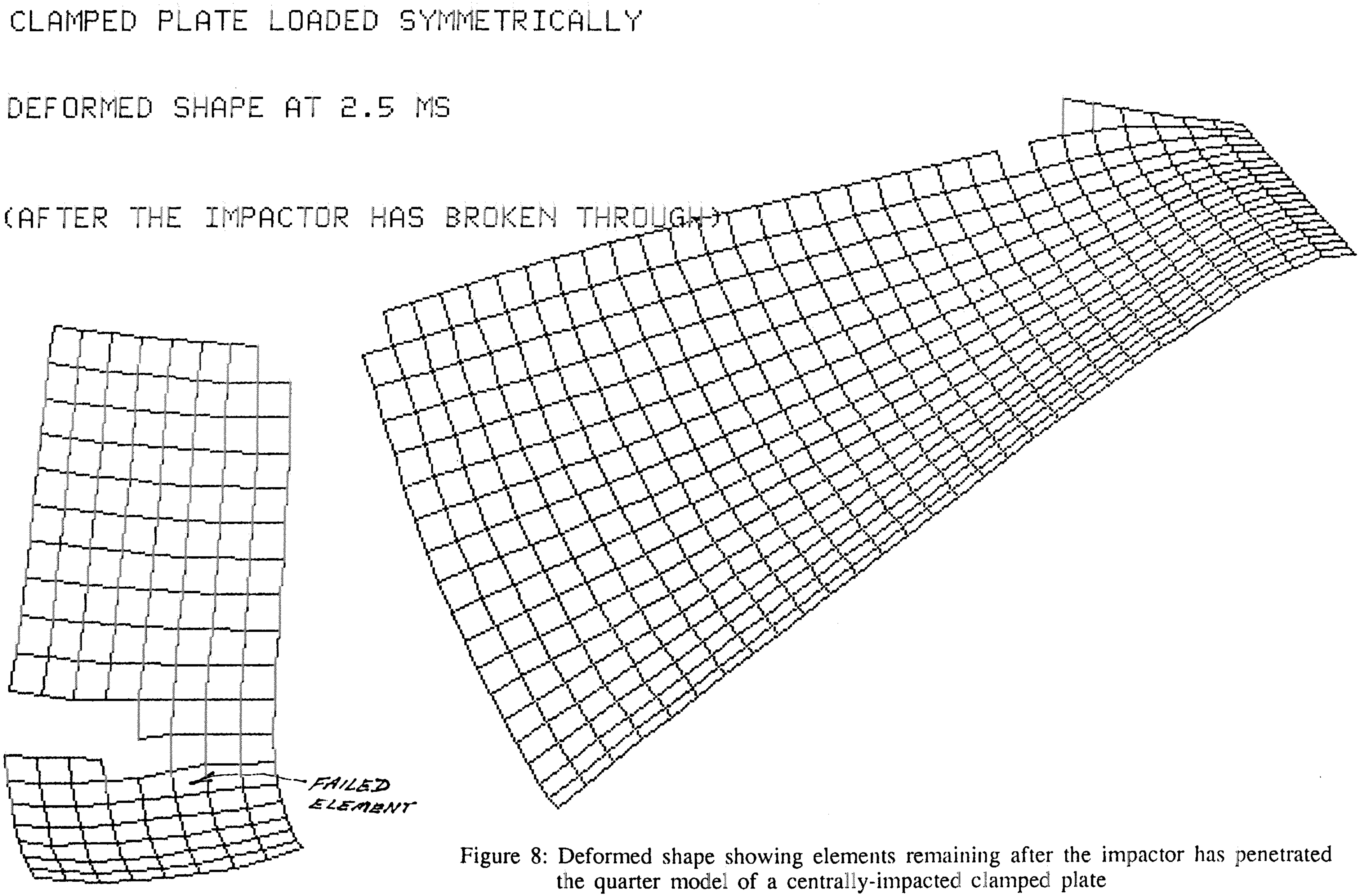


Figure 8: Deformed shape showing elements remaining after the impactor has penetrated the quarter model of a centrally-impacted clamped plate

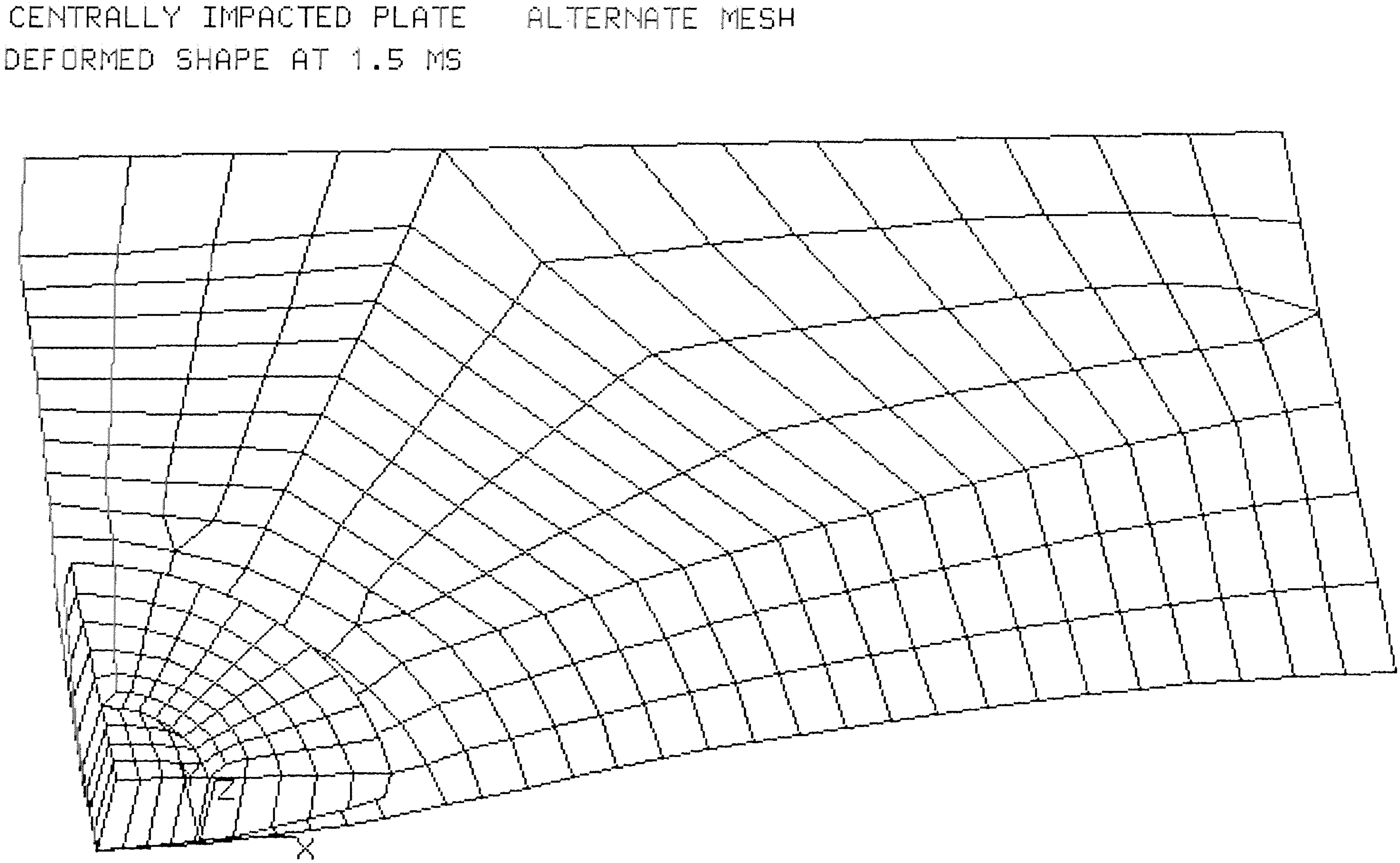


Figure 9: Alternate quarter model of a centrally impacted plate

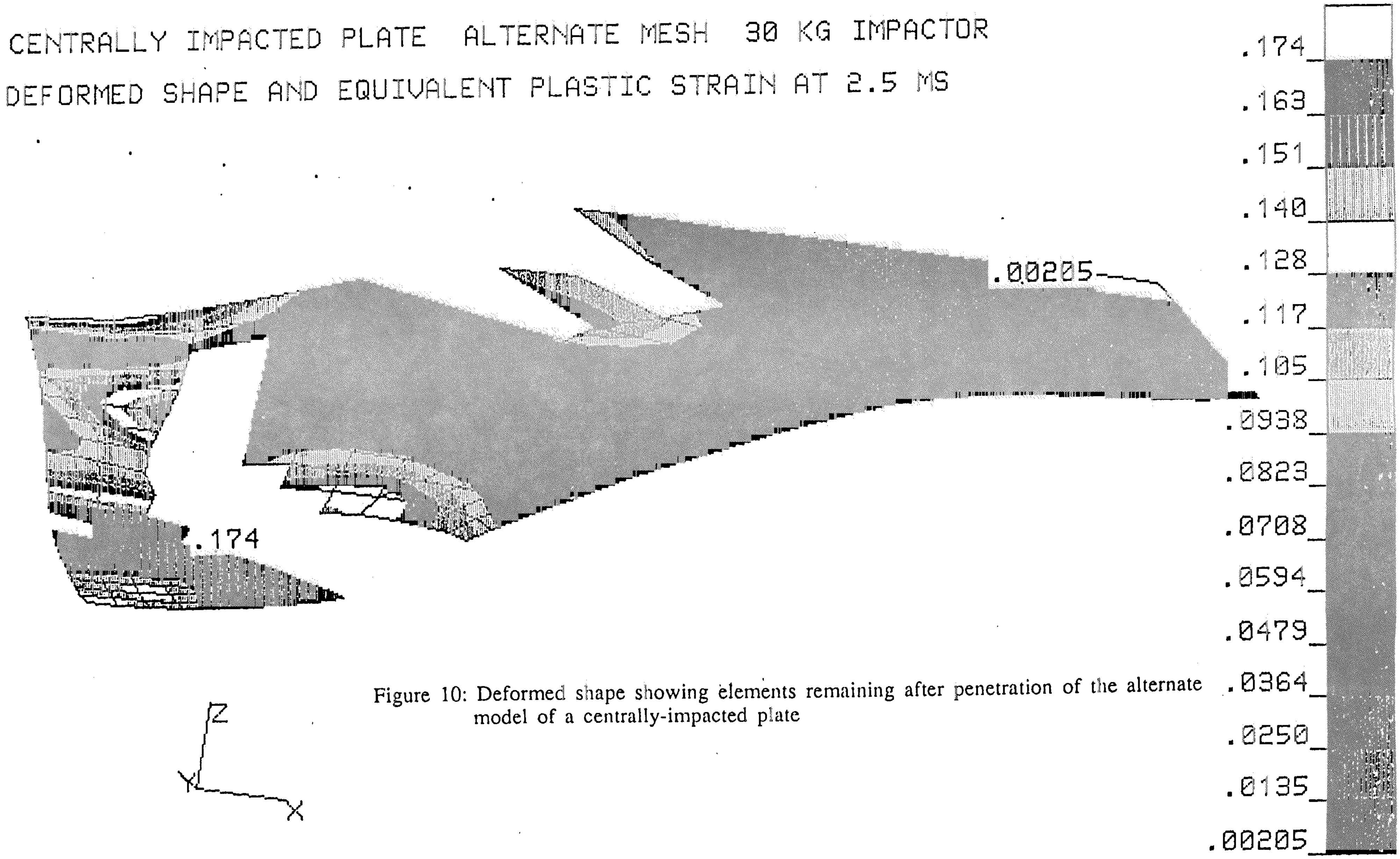


Figure 10: Deformed shape showing elements remaining after penetration of the alternate model of a centrally-impacted plate

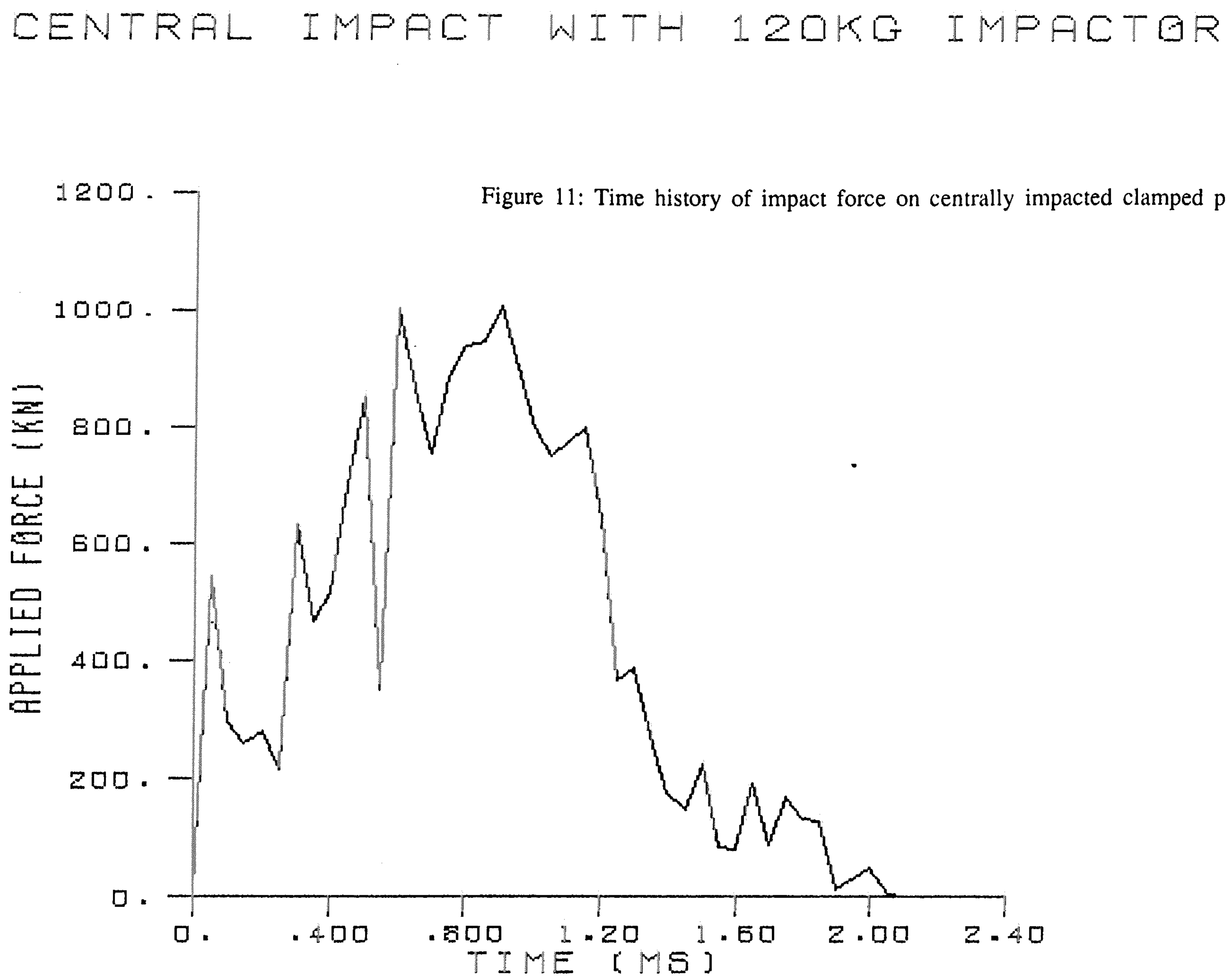


Figure 11: Time history of impact force on centrally impacted clamped plate

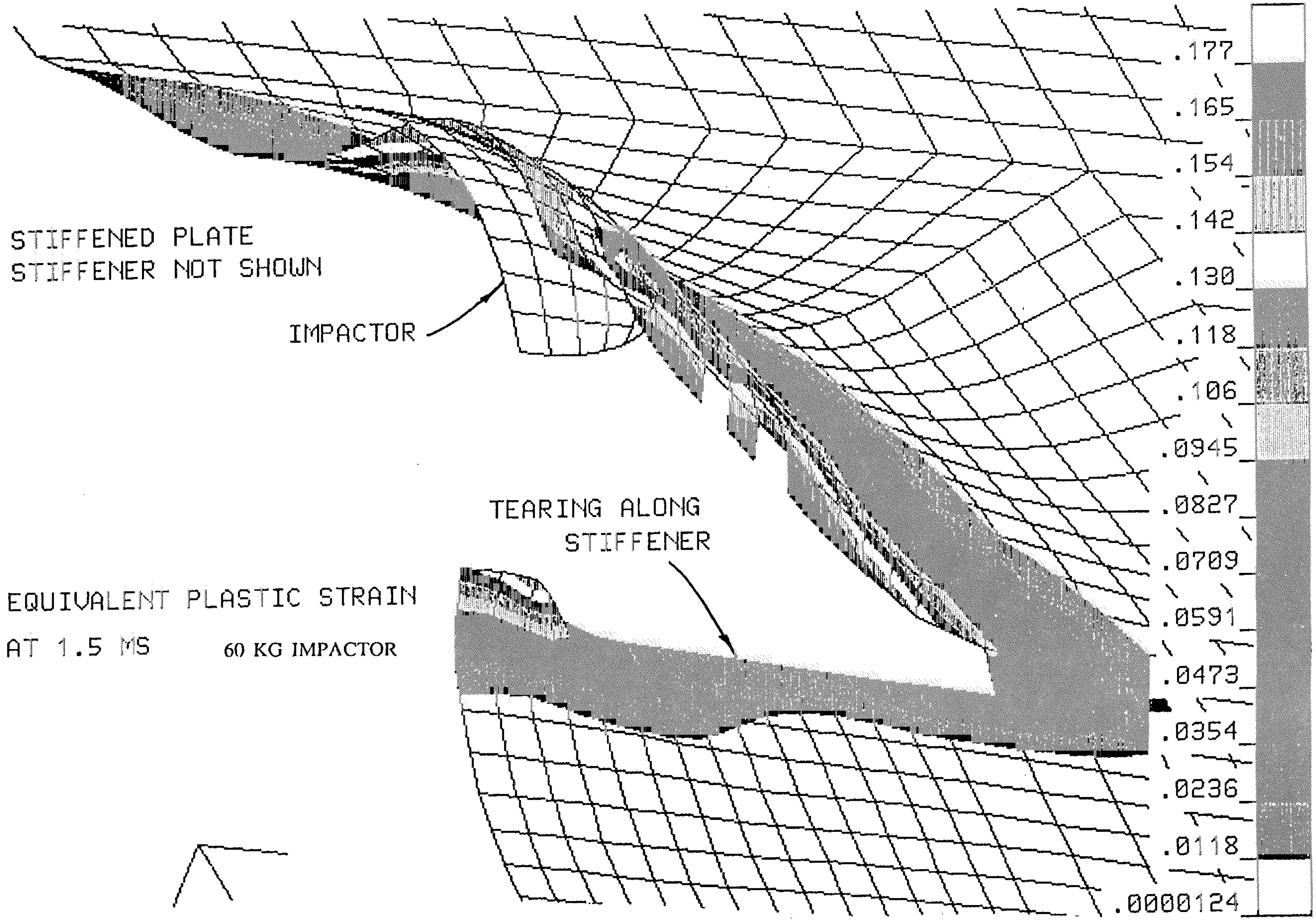


Figure 12: Deformed shape of stiffened plate, showing tearing as impactor penetrates

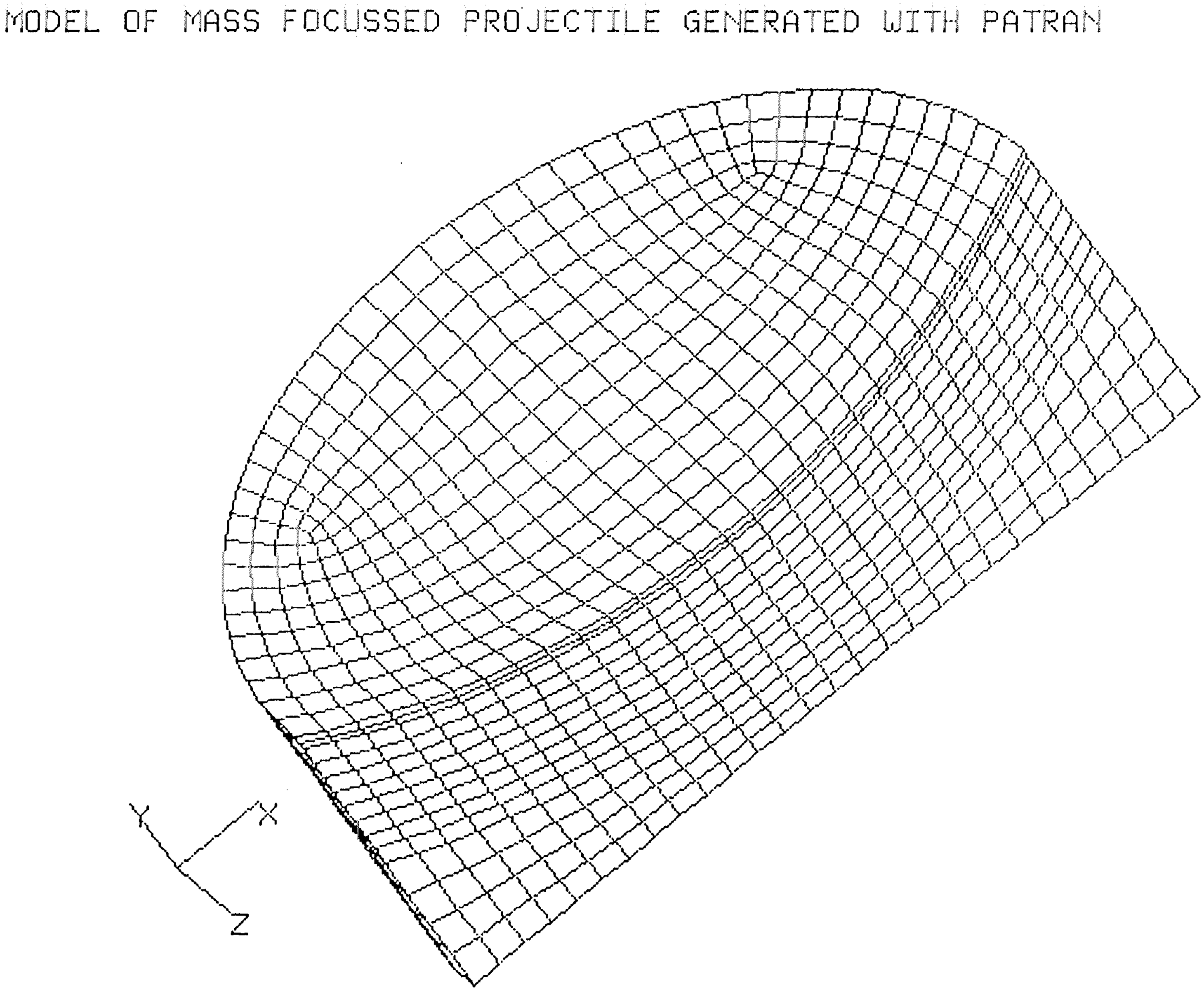


Figure 13: Half model of copper disk resting on explosive material

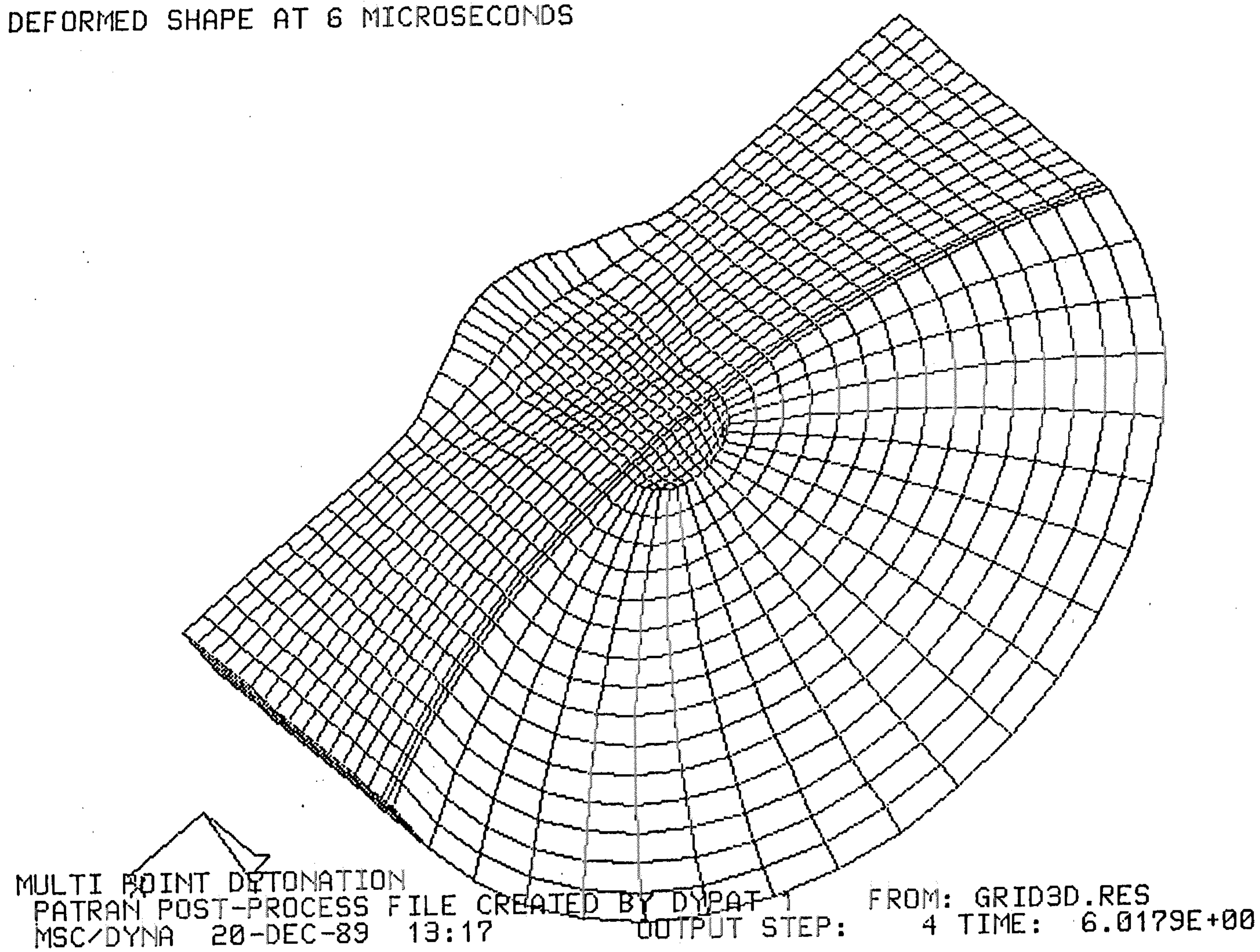


Figure 14: Shock wave propagating through an explosive material

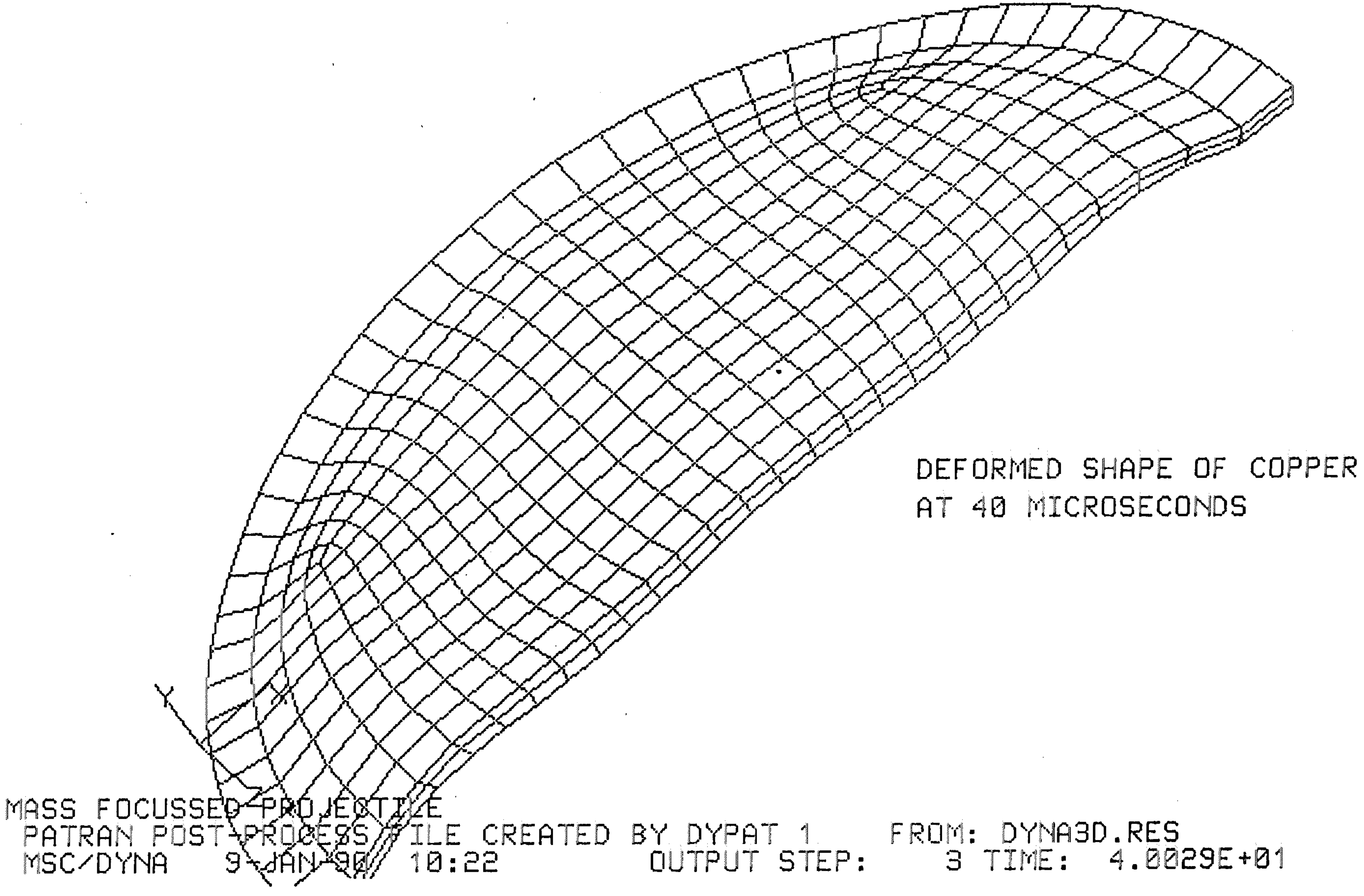


Figure 15: Deformation of the explosively formed copper disk at 40 microseconds

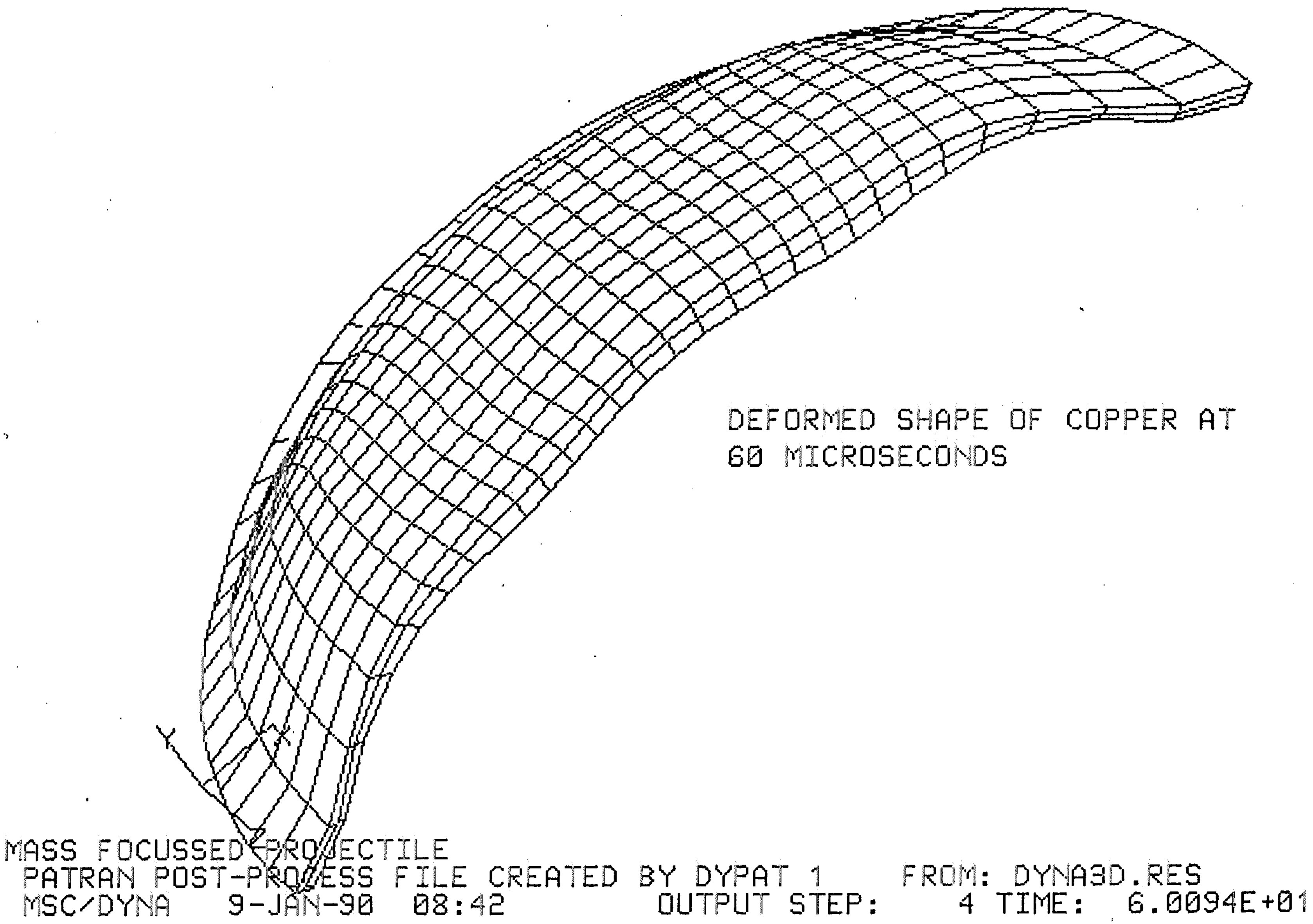


Figure 16: Deformation of the explosively formed copper disk at 60 microseconds

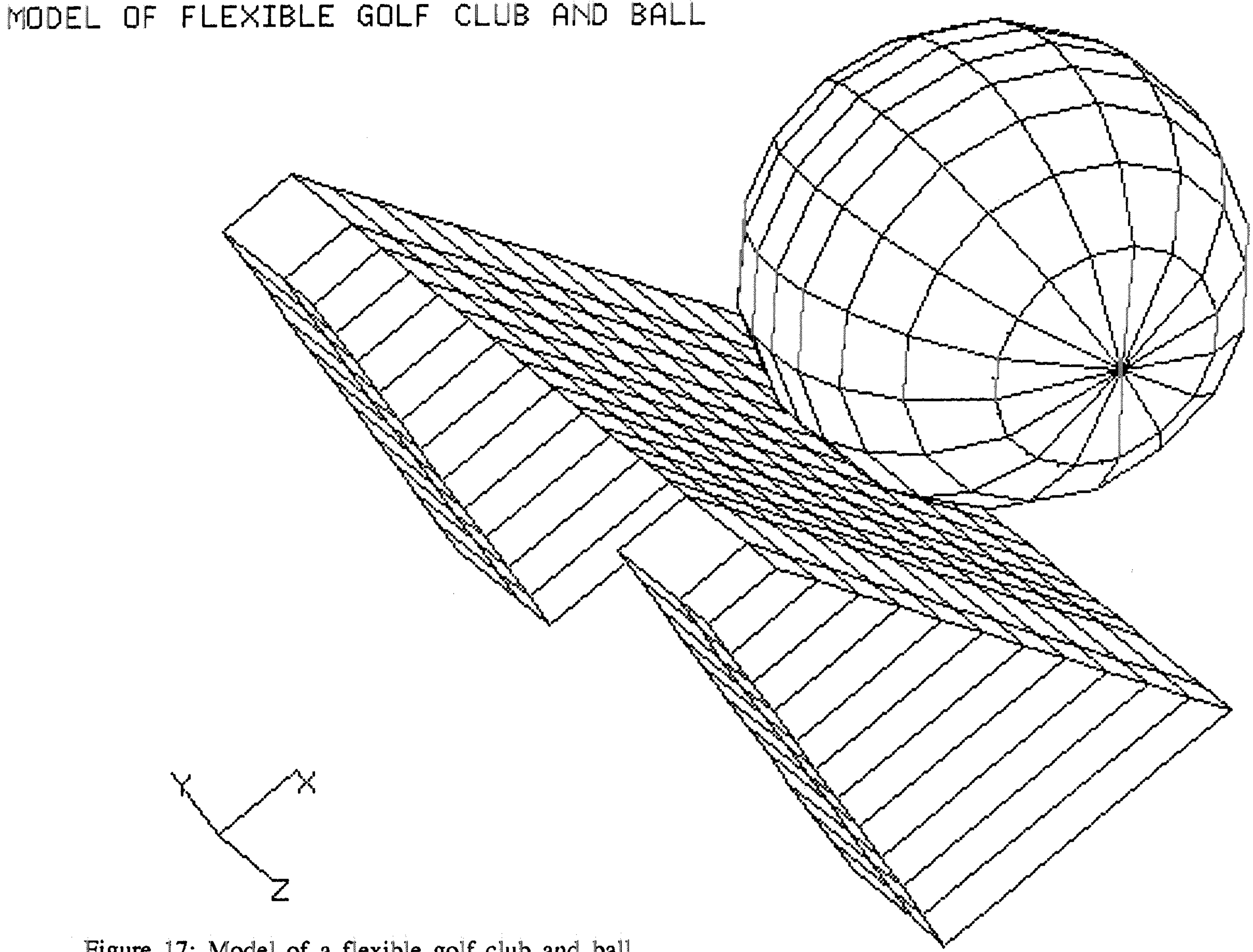


Figure 17: Model of a flexible golf club and ball

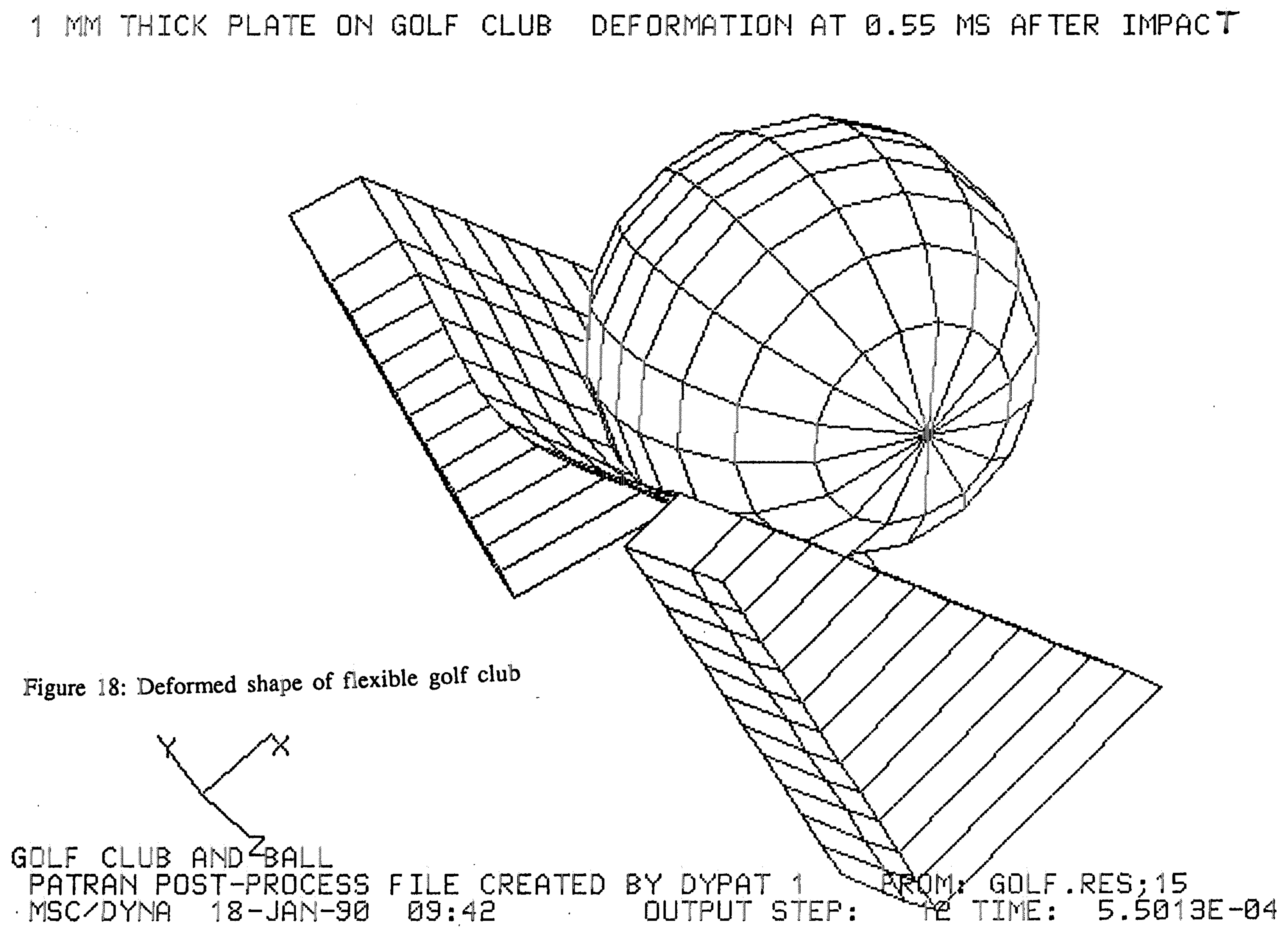


Figure 18: Deformed shape of flexible golf club

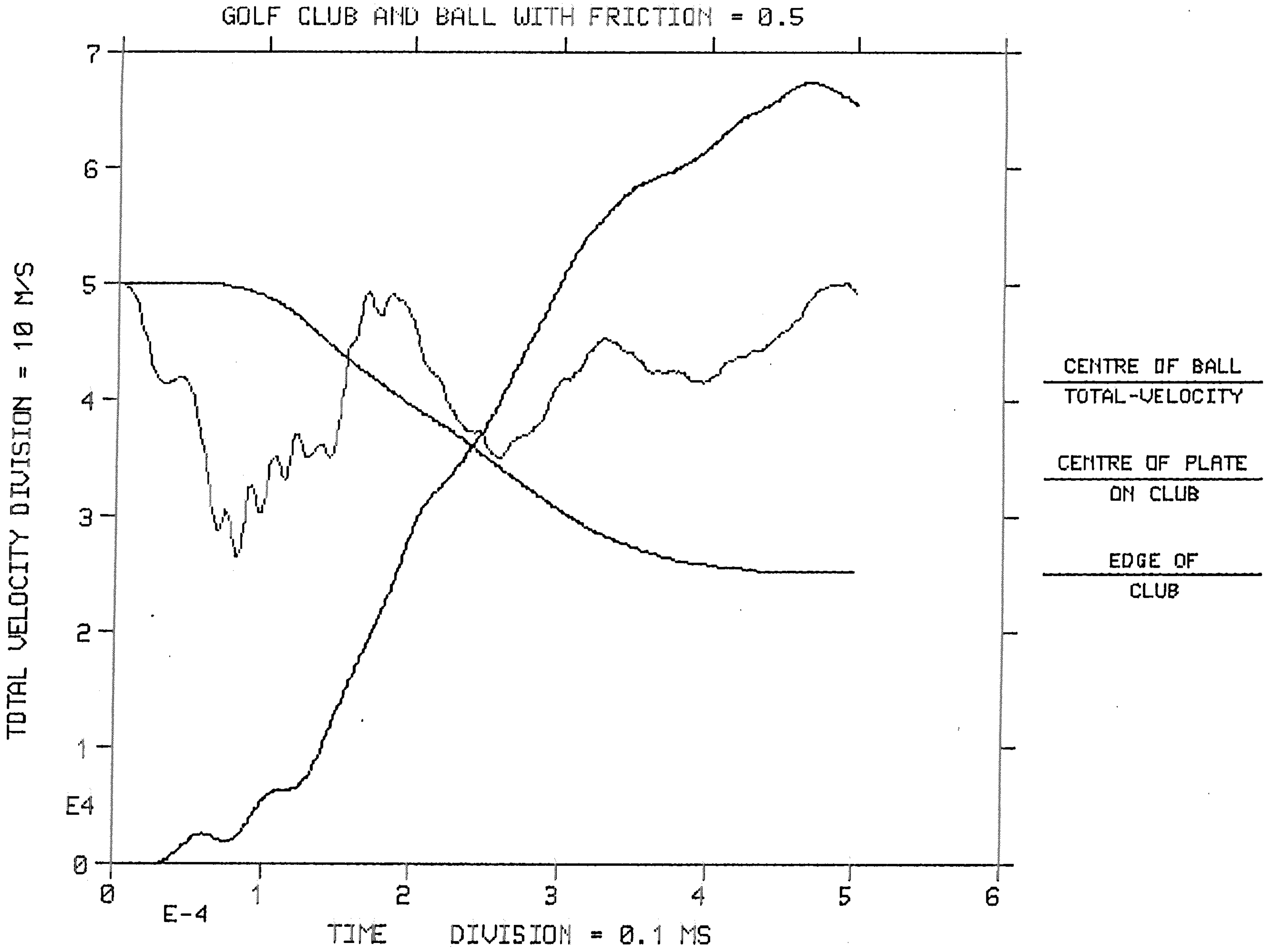


Figure 19: Time history of velocity of golf club and ball while in contact

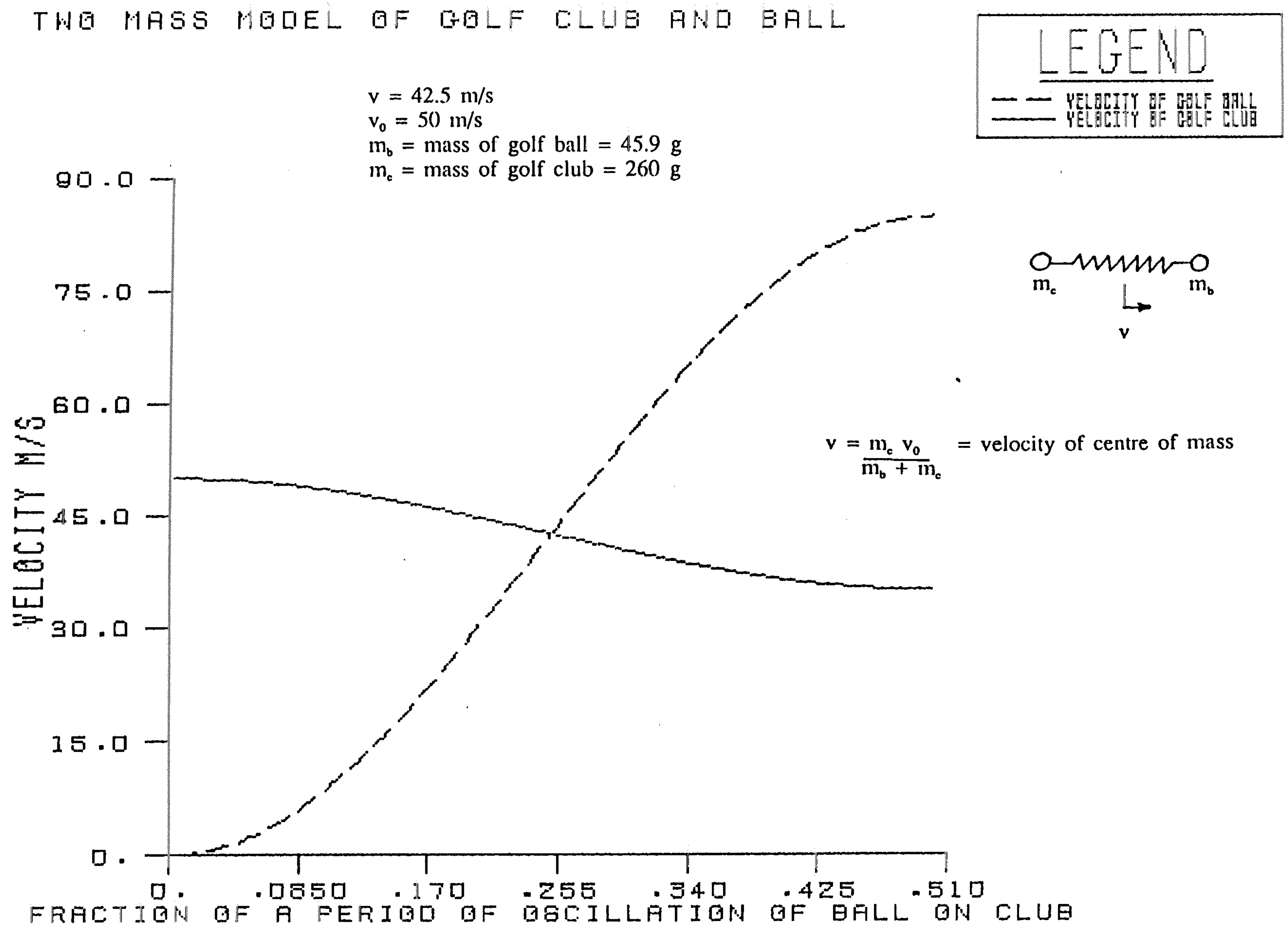


Figure 20: Theoretical response of a two mass system of golf club and ball