COMPUTER SIMULATION OF ELASTIC DEFORMATION ON CAR FRAMES FOR HYDRAULIC LIFTS

Enrico M. Penati
I.G.V. S.p.A.
via Pacinotti 4
20090 Segrate - Milano - Italy

ABSTRACT

The introducing and the spreading of computers have made possible, within structural engineering, the removal of the calculation difficulties inside the design phase of structures having a high degree of super-staticity .
The accessibility and the easiness of use of the evoluted languages of programming allow also the creation and the personalisation of the relevant calculation codes .
This report describes an example related to the lift technique (analysis of the internal action and of the deformation of the frame holding the car).
The numerical algorisms used in the linear elastic area, and the possibility of their interfacing with other applicable programs are also mentioned .

1) INTRODUCTION

In the last years we have seen P.C. and mini computers becoming more and more numerous especially in most of the technical department : this happened both for their low price and for the high number of available software packages .
All the engineering specializations, today, can be supported by informatics : drawings, calculations, data recording, etc.
Of course, it's impossible believing this can replace engineer's experience and capability, but is clear that computers can relieve repetetive activities and let the man choosing, in a short time, many different solution for the same problem .
Most of all, this is true if we consider structural engineering, a professional field where, since more than 10 years, a new tool is available : Finite Elements Method (FEM) .

2) WHAT IS FEM ?

Finite Elements Method is a numerical technique which emulates the differential equations associated with static and dynamic equilibrium of a continuous system ; this is performed by using an algebraic equations system with a finite number of unknown values
The first step is modelling the continuous : it's possible to use triangular or quadrangular elements to study plates and shells, beams and trusses to study three-dimensional frames, and so on .
The mahematical model built in this way can be represented by the following vector equation :

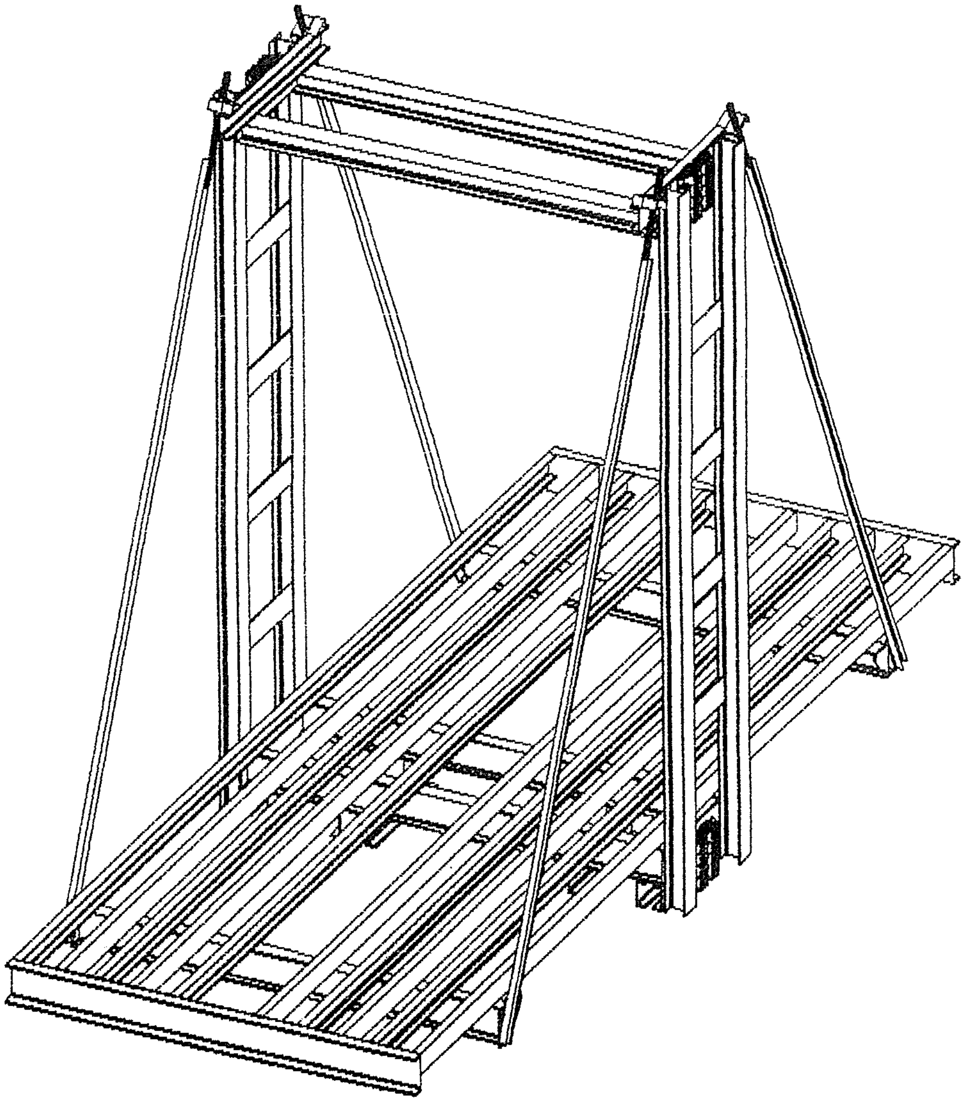
[R] = [K] [U]

where [K] is the stiffness matrix, which synthesizes the elastic properties of the material, the geometric dimensions of the elements and their assembling ; [U] is the vector of resulting displacements ; [R] is the vector of applied loads .
We can see the logical advantages of such a method :
a) it's an accurate method and it's as much closer to the reality as much the meshes are more numerous ;
b) it's an optimized way to proceed, because, keeping to generalities, matrix [K] inversion (it must be made just once) allows to emulate different load conditions and combinations .

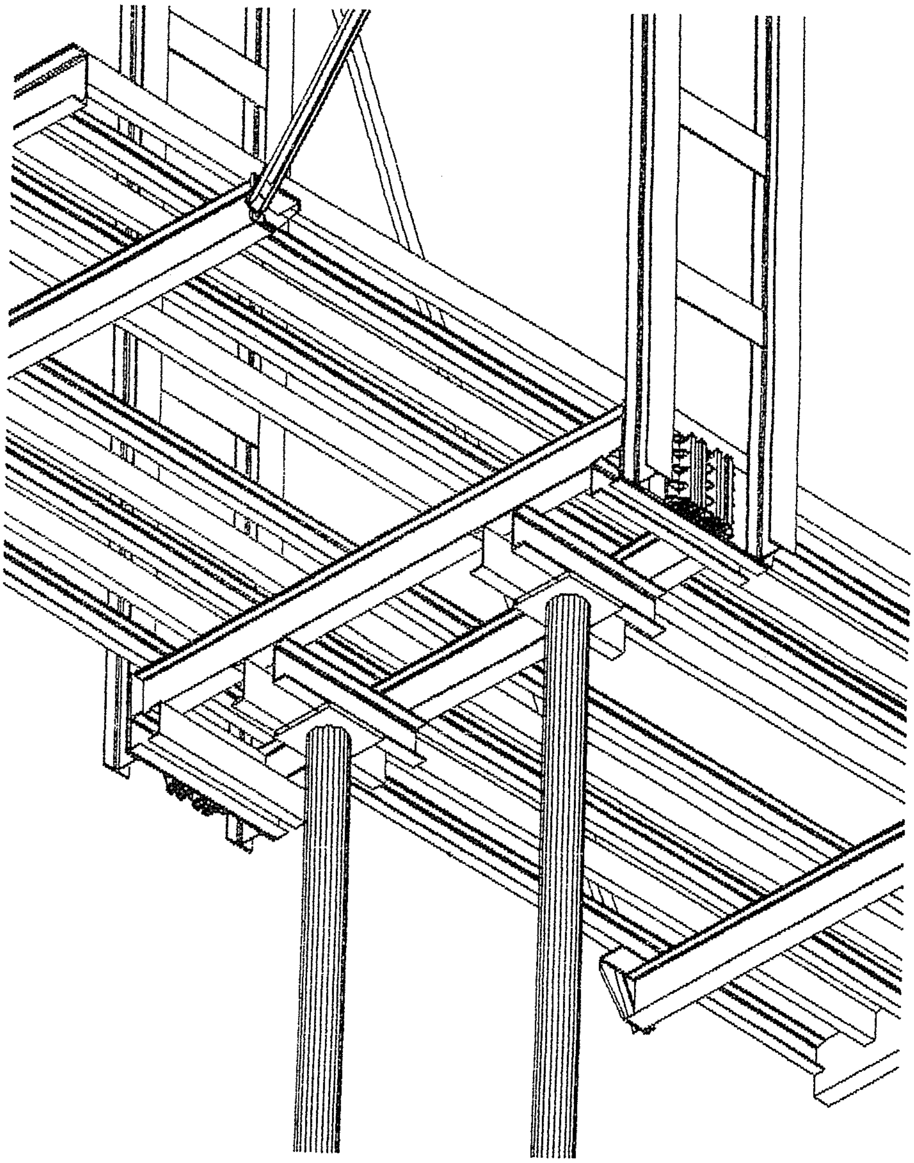
But it's obvious that using this method was made really possible only when proper hardware was available and ready for handling big dimension matrix (3000 equations or more) .
In the structural anlysis, at the present, old iterative methods (Cross, Grinter, etc.) are completely replaced by FEM .
More, made possible to study any type of field-problem (heat transfer, acoustics, etc.) .

3) HOW TO WORK WITH FEM ?

As far as three-dimensional frames are concerned, it needs to describe number and position of the joints, creating the nodal and truss (or beams) coordinate data, type of sections, material properties, degrees of freedom and applied loads .
It's often a boring and can cause some mistakes if you don't pay much attention to it : however, there are a few solution that can help us .
One of these is exchanging data with a CAD system : while working drawing is going on, it's possible to decribe the static model of the frame structure, reducing the event of mistakes .
You can see a few examples of this in the figures : a car frame description made by CAD (with schematic joints and beams) ; this was produced at the same time of the other complete drawings you can see .
Data exchanging was possible using entities (attributes) associated to the drawing and then translating them into ASCII files, read by the FEM program .



CAD generated isometric view of the car frame from above



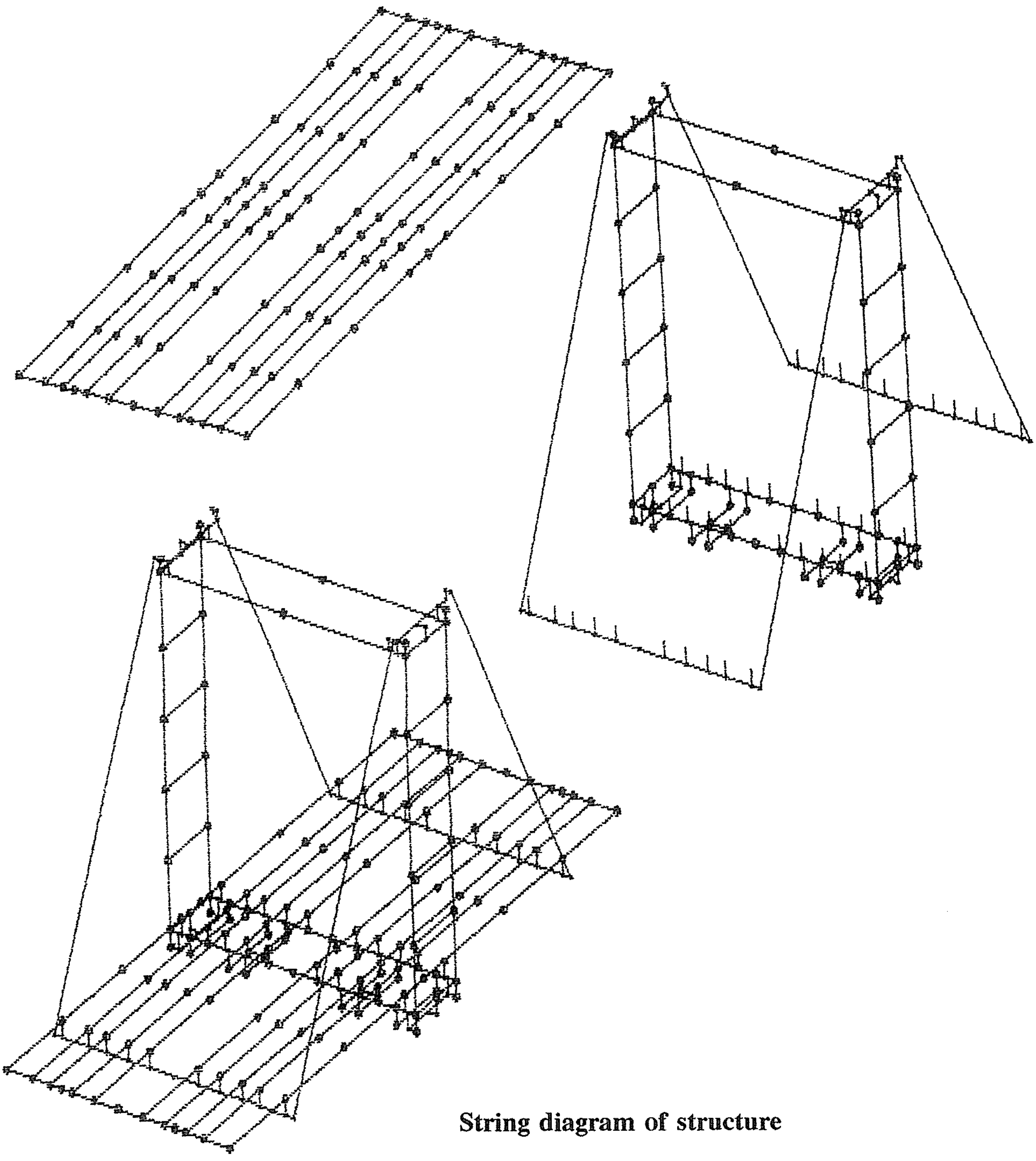
CAD generated isometric view from the pit

4) A REAL EXAMPLE

As we already told you, we have studied (and built) a car frame for a hydraulic lift, with the following features .

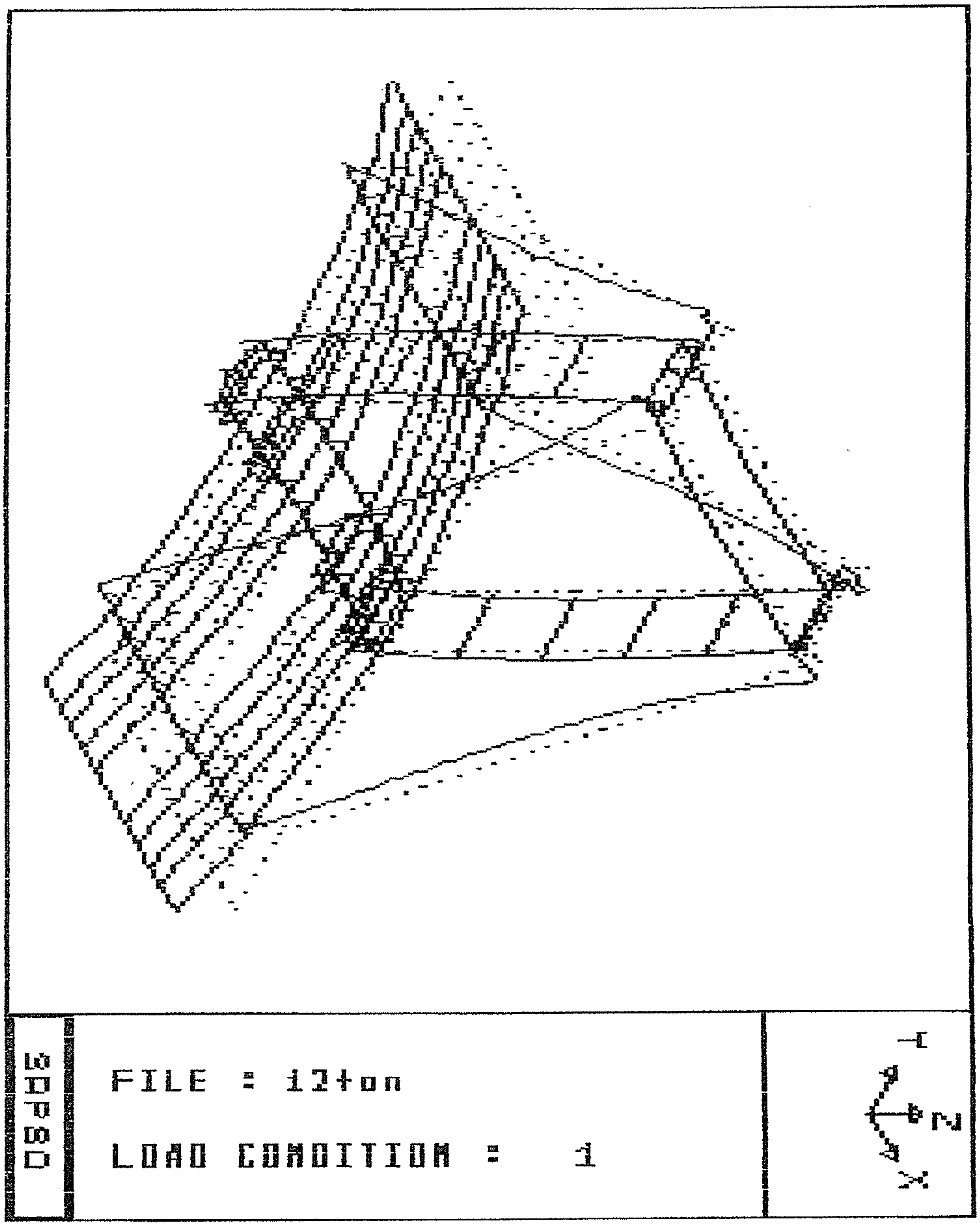
Rated load (trucks)	:	12000 [Kg]
Speed	:	0.15 [m/sec]
Travel	:	8.9 [m]
Rams	:	n 2 ; dia. 200 [mm]
Working pressure	:	16-35 [bar]
Total power installed	:	56 [KW]
Car inside dimensions	:	3000 x 4200 x 8000 [mm]

Both rams were placed near the center of the car, on the ideal axis of guides : this choice was made to avoid bending or jibbing effects for the rams due to eccentric load position, guides and car frame deformation, etc.
The problem was reducing total car and car frame mass to less than 10000 Kg., limiting as much as possible bending deflection at the edge of the car due to truck coming in .



String diagram of structure

Structure desciption was made using 269 joints and 354 beams .
The following load conditions and combinations were evaluated :
a) weight of car and carframe
b) rated load concentraded on 4 points
c) ecentric load according two direction (width and length of the car)
In all the above mentioned conditions, displacements are satisfactory .
We want to outline how this analisys takes into consideration both shear and twisting effects and also secondary bendings in the nodes .
It's possible to see a plotted deformed shape of the car frame in the figure .



5) CONCLUSIONS

Short execution time, easy possibility to change loads and degrees of freedom and the consequent evaluation of different solutions are basical for structural analisys .
Moreover the great need of working cost reduction oblige to optimize structural design (less material and easy assembly) respecting the reliability .
For this reason automatic calculation codes are getting more and more common in engineering .